Crusader II Troubleshooting Manual
70000253 @ $50

ANILAM

Anilam Electronics Corporation

CRUSADER II
PROGRAMMING MANUAL

PART NO. 700-019 D

SAFETY NOTICE

FOR CLARITY PURPOSES, THE DRAWINGS IN THIS MANUAL HAVE BEEN ILLUSTRATED WITHOUT PROTECTIVE GUARDS IN PLACE.
WHEN APPLYING THIS CONTROL IN A RETROFIT APPLICATION YOU ARE CONVERTING A MANUALLY CONTROLLED MACHINE. CONSIDERATION MUST BE GIVEN TO MEETING APPLICABLE SAFETY AND GUARDING STANDARDS. THESE STANDARDS CAN BE OBTAINED FROM:

1. "Safety Requirements for the Construction, Care and Use of Drilling, Milling, and Boring Machines", ANSI Standards Institute, 1430 Broadway, New York, New York 10018.
2. "Point of Operation Guarding" Section 1910.212, U.S. Occupational Safety and Health Administration
3. "Power Transmission Guarding" Section 1910.219, U.S. Occupational Safety and Health Administration both of which are available from the U.S. Department of Labor.

(Information in this manual subject to change without notice.)

PART NO. 700-019D
PRINTED DEC. 1984

BASIC SPECIFICATIONS

NO.	ITEM	SPECIFICATION
1.	Controlled Axes	Three axes (X, Y & Z)
2.	Simultaneously Controllable Axes	Any two, or all three, in rapid, feed, & manual operation
3.	Increment System	Input: X, Y, Z .0001 inch .01 mm Output: X, Y, Z .0005 inch .01 mm
4.	Maximum Programmable Dimensions	± 999.9999 inches ± 99999.99 mm
5.	Decimal Point Programming	Data can be input with a decimal point.
6.	Rapid Traverse Rate	Rapid traverse is at 100 inches per minute (IPM).
7.	Feed Rate	Feed rates from 80 to .1 inches per minute are programmable. An override of 0% to 120% (5% increments) is provided.
8.	Absolute/Incremental Programming	Axis moves may be absolute or incremental, but may not be mixed in the same event.
9.	Axis Zeroing	Any axis may be zeroed at its present location, or preset to any dimension.
10.	Positioning in Rapid	A Rapid command moves the machine @ rapid traverse to the position specified. The machine deaccelerates to a stop, and an in position check is made before execution continues.
11.	Linear Interpolation	A straight line move is made to the next X-Y, X-Z, or Y-Z position @ the feedrate programmed.
12.	Circular Interpolation	Any arc between 0 & 360° may be programmed, CW or CCW.
13.	Dwell	Execution of a dwell command stops program running until the start button is pressed. A timed dwell will automatically restart. Time may be .1 to 999.9 secs.
14.	Canned Cycles (G-codes)	A variety of miscellaneous canned cycles are accessed by a 2-digit G code.
15.	Auxiliary (Aux-codes)	Various features (mirror image, spindle control, etc.) are activated by a 4-digit aux code.
16.	Tool Function	Tool offsets are called by a 2-digit tool number (01 thru 32). Tool offsets are set by a 4-digit number (1001 thru 1032).
17.	Hold	The movement of all axes may be stopped and resumed by using the hold & start button.
18.	Single Step	Commands may be executed event by event or move by move.
19.	Emergency Stop	When the emergency stop button is pushed, all servo motors & spindle motors stop.

20.	Overtravel	All axes should be equipped with limit switches, which will shut down all servos & spindle motors. Machine must be moved off limit switch before servos will restart.
21.	Manual Feed or Rapid Moves	Moves may be made by holding down the appropriate key.
22.	Jog Moves	Positioning in incremental amounts can be performed as follows: .001, .010, .100 (inch) .01, .10, 1.0 (mm)
23.	Do Loop	A sequence of events may be repeated from 0 to 999 times.
24.	Subroutines	A subroutine may call another subroutine to a nested depth of 32 levels.
25.	Program Protection	10 minute battery back-up of program and position with ability to use tape cassette.

OPTIONS

1.	Line Voltage	110 VAC or 220 VAC.
2.	Advanced System I Software	Provides many additional canned cycles & features.
3.	RS 232	Links control to other peripherals.
4.	Program Saver	Tape cassette storage & replay of programs.

INTRODUCTION

This guide should be used by the machine operator to learn the programming format for the Crusader I or II CNC Microprocessor Control system. It also will serve as a quick reference for CNC programmers that wish to familiarize themselves with the Crusader.

This manual is designed to teach. In order to use it effectively, the instructions and examples must be followed in step-by-step order. Understand each section before going on to the next section. This approach is necessary to thoroughly familiarize you with the Crusader and its functions.

This manual will first explain important points of axis direction and dimensioning theory. It is necessary to understand these subjects before using any CNC.

This manual will then define the functions of each button on the programming keyboard. References are also given so the operator may see each button in use within the many working examples later included in the manual.

Section 5 prepares the operator for the programming of a simple part. It includes a section on program editing and ends with the operator actually dry-running a part with the Crusader control.

Sections 6 - 10 will take the operator through several practical, "hands-on" programming examples. At the same time, the operator will learn about various Crusader features such as tool length offsets, mirror image, do loops and cutter diameter compensation, to name a few.

Finally, the Advanced Software System-1 package and its features are fully detailed in the last chapter. This optional package enables the operator to greatly reduce programming time through the use of canned cycles.

A review of the Table of Contents will give you a concise picture of the Crusader's capabilities.

Bear in mind that the Crusader control is programmable and will follow all instructions exactly. If unsafe or incorrect instructions are given to the Control, it will follow these instructions.

One last note; ensure that you understand the use of the **Emergency Stop** button to avoid damage or injury. Do not hesitate to use Emergency Stop if a dangerous or unsafe condition arises.

CRUSADER PROGRAMMING MANUAL

INSIDE COVER STATEMENT: SAFETY NOTICE

CONTROL SPECIFICATIONS
INTRODUCTION
TABLE OF CONTENTS

SECTION 1: CNC ORIENTATION AND START UP
- A. Power Supply 2
- B. Turning the System on 2
- C. Using the Machine Manually 2

SECTION 2: PART DIMENSIONING AND THEORY
- A. Axis Direction 4
- B. Absolute Dimensions 4
- C. Incremental Dimensions 5

SECTION 3: VARIOUS MODES OF OPERATION
- A. Starting the Servo Motors 8
- B. Manual 8
- C. Setting Absolute Zero 8
- D. Program Enter 8
- E. Program Check 9
- F. Single Step 9
- G. Auto 9
- H. Feed % 9
- I. Inch-millimeter (in/mm) 9

SECTION 4: PROGRAMMING KEYBOARD DEFINITIONS
- A. Disp Posn (Display Position) 11
- B. Do (Do Loops) 11
- C. Sub (Subroutines) 11
- D. Call 11
- E. End 12
- F. Rapid/Feed 12
- G. Incr (Incremental) 12
- H. Dwell 12
- I. Aux (Auxiliary Codes) 12
- J. Tool (Tool Length Offsets) 12
- K. Arc 12
- L. CW/CCW (Clockwise/Counter-clockwise) 12
- M. G (G codes) 13
- N. V (Variables) 13

SECTION 5: PREPARATION FOR PROGRAMMING
- A. Holding the Workpiece (Step 1) 16
- B. Determining a Reference Point (Step 2) 16
- C. Tool Selection (Step 3) 16
- D. Beginning a Program 17
- E. Clearing the Memory (Event Clear) 18
- F. Program Entry (Event Enter) 18
- G. Editing (Event Search) 18
- H. Program Execution 19
- I. Circular Motion 19

SECTION 6: PROGRAMMING EXAMPLE (DRILLING AND SPOT DRILLING)
A. Explanation of Events (Example 1) 22
B. Entering the Program 23
C. Add Event 25
D. Delete Event 25
E. Machine Set-up 25
F. Tool Length Offsets 25
G. Review of Program Format 26
H. Running the Program 26

SECTION 7: DRILLING EXAMPLES
A. Drilling Format Rules 28
B. Example 2: Do Loops 29
C. Example 3: Subroutines 30
D. Nesting 31
E. Example 4: Peck Drilling 32
F. Example 5: Mirror Image 33

SECTION 8: MILLING EXAMPLES
A. Example 6: Circular Milling 36
B. Example 7: Mirror Image 37
C. Example 8: X Z Contouring 37

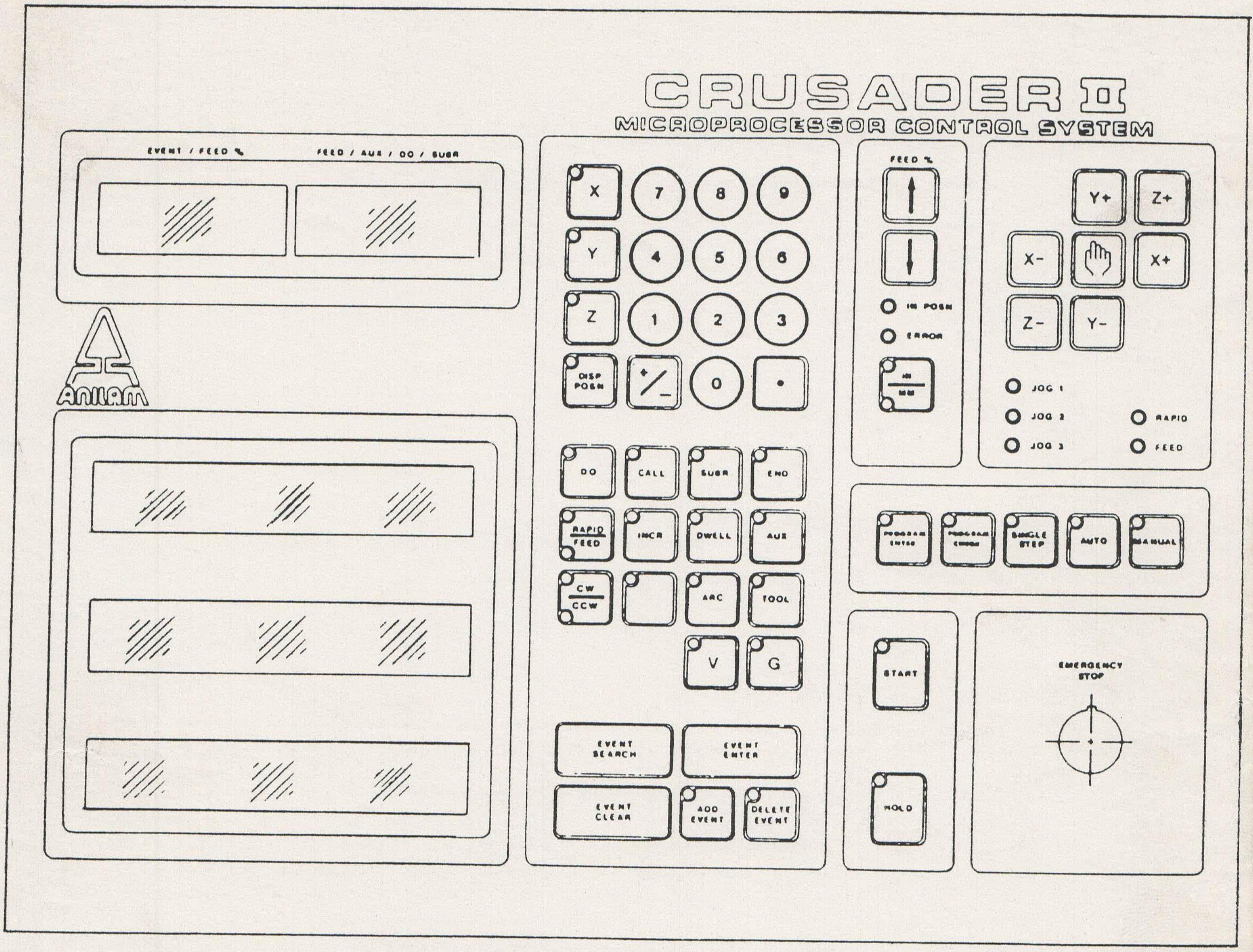
SECTION 9: ADDITIONAL FEATURES
A. Cutter Diameter Compensation 40
B. Example 9 44
C. Example 10 45

SECTION 10: MISCELLANEOUS
A. Auxiliary Code Listing 48
B. Tape Cassette 50
C. Error Light 51
D. Presetting Zero 51
E. Manual Programming Moves 52
F. Programming from the Part 52
G. Feedrate Compensation 52
H. Diagnostics 53
I. Crusader Guidelines 54
J. Center Drill Chart 55
K. Common Tool Point Formulas 56

SECTION 11: ADVANCED SOFTWARE SYSTEM - ONE
A. Polar Coordinates G10 - G13 58
B. Polar Rotation G51, G52 60
C. Scaling G53, G54 62
D¹ Frame Milling Cycle 62
D. Hole Milling G76 63
E. Circular Pocket Milling G77 63
F. Rectangular Pocket Milling G78 64
G. Bolt Circle G79 65
H. Absolute Zero Shift 66
I. Software Limits 66
J. Helical Interpolation 67
K. V Code Format Rules 68

SECTION 12: ADVANCED SOFTWARE SYSTEM - TWO

A. Elliptical Interpolation G05 71
B. Spiral Milling: Planar or Helical G06 71
C. Conical Milling Cycle G48 72
D. Radial Conical Milling Cycle G49 72
E. Mold Rotation Cycles G45 - G47 73
F. Cancelling Plane Rotation AUX 9030 74
G. Plane Rotation AUX 9031 - 9033 74
H. Drilling Canned Cycles G80 - G89 75



Crusader Control (faceplate) with Variable Key

Figure 3

SECTION 1:
CNC ORIENTATION AND START UP

CAUTION:

DO NOT attempt to turn on the console or machine until a QUALIFIED service technician has checked the system. Serious damage may result if this is attempted. The machine and control must be in place and checked before operation.

A. POWER SUPPLY

Power is supplied to the system thru a 115 volt (230 volt optional) power supply cable. This is attached to the servo amplifier cabinet usually located on the right side of the machine column as shown in figure 1. Power must be constant at a minimum of 105 volts AC and it must be properly grounded. The power cord can now be plugged in to an AC power source.

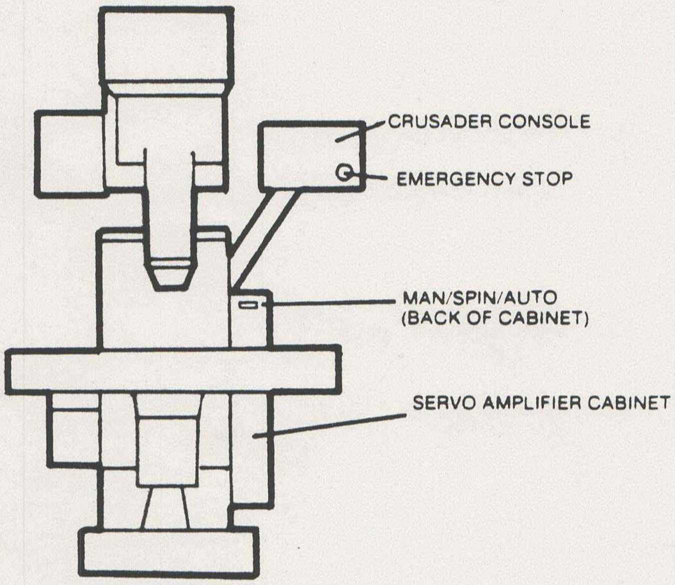


Figure 1

B. TURNING THE SYSTEM ON

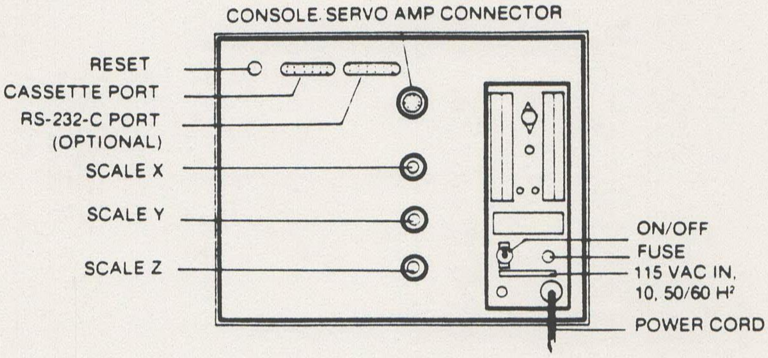
To turn the system on, the console must first be started. Do this by placing the ON—OFF toggle switch, located on the back of the Crusader console, in the ON position, (see figure 2). Now, push the emergency stop button in and then pull it out to reset the console.

When the toggle switch is turned on, the keyboard and faceplate should look exactly as described here:* The **HOLD** button will not be lit if the **EMERGENCY STOP** button was pulled out before the toggle switch was turned on. Depending on how the console was set up at the factory, the inch (IN) or metric (MM) button will be lit.

***The buttons which should be lit are MANUAL, INCH, HOLD, DISPLAY POSITION, RAPID, and IN POSITION. The digits 1.00 will show in the Feed % display and all "O" 's will show in the axis displays.**

If some lights are off or on which should not be, or the digits are only partially lit, turn the console off and call your nearest Anilam distributor or the Anilam service line at 1-800-327-6340.

If no lights at all appear, refer to the spare fuse listing. Spare fuses are in the servo amplifier box in a pocket attached to the inside of the cabinet door.



CRUSADER II CONSOLE (REAR VIEW)

Figure 2

C. USING THE MACHINE MANUALLY

On the back of the servo amp cabinet (see figure 1) is the Spindle toggle switch marked **MANUAL AUTO**. Put this in the **MANUAL** position. This is the position the switch will be in when the machine is to be used with the hand wheels without the use of the servo motors.

To start the spindle, put the toggle switch mentioned above in the manual position, push in, twist, and pull out the **EMERGENCY STOP.** Press the **RESET** button on the back of the console, then turn the spindle motor on with its original switch. The machine can now be used as a manual mill or for set up.

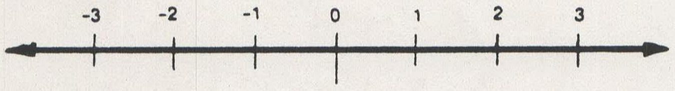
Use the machine in this mode while reading Section 2. (Part Dimensioning and Theory).

SECTION 2:
PART DIMENSIONING AND THEORY

A. AXIS DIRECTION

Figure 4 shows a simple number line. It has a reference point zero, which is called absolute zero, and is divided into increments. Moving away from absolute zero, to the right, are positive numbers. Positive numbers have no (+) sign. If a number does not have a (-) sign it is a positive number. Moving from absolute zero, to the left, are negative numbers. Use the plus or minus sign of a number to indicate its relationship to absolute zero. To name a point 3 increments to the left of absolute zero, use the term -3. absolute.

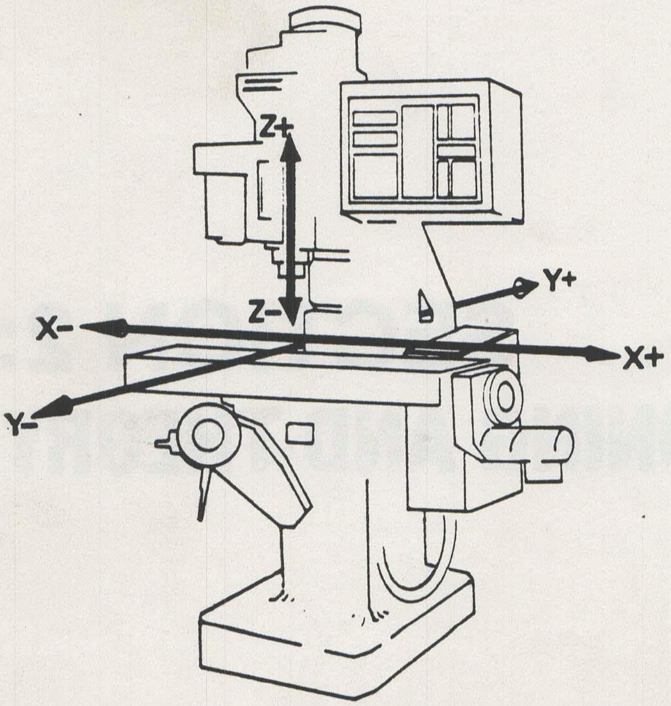
Imagine a number line that is placed along each axis of the mill that we can control. The increments of the number line are .0005 (.01mm). An absolute zero can be set for each number line. Once an absolute zero is established, use that reference to move the machine in a plus or minus direction along that number line.



NUMBER LINE

Figure 4

Always think of the direction that the cutting tool (spindle) is moving. The plus or minus direction is the direction of the cutting tool, not the table.



AXIS DIRECTIONS

Figure 5

From the normal operator's position in front of the mill, when the cutting tool moves from absolute zero to the right we will refer to this as an X+ absolute dimension, to the left of zero as X- (figure 5). When the tool moves from absolute zero away from the operator towards the column, it is a Y+ dimension, towards the operator is Y-. As the quill moves up from absolute zero it is Z+, down from zero is Z-. Move the machine with the handles and the numbers in the displays will correspond to the tool direction.

Do not proceed until all axis directions and plus and minus movements are understood. This section (Axis Direction) will be the reference for all programmed motion.

B. ABSOLUTE DIMENSIONS

Figure 6 shows the X and Y axis viewed as if you were looking down at the table of the machine from above. Absolute zero can be set at any point on the table (Setting absolute zero will be covered later.) At this time, absolute zero will be at the position where the center of the spindle was when the console was turned on.

QUADRANT DIAGRAM

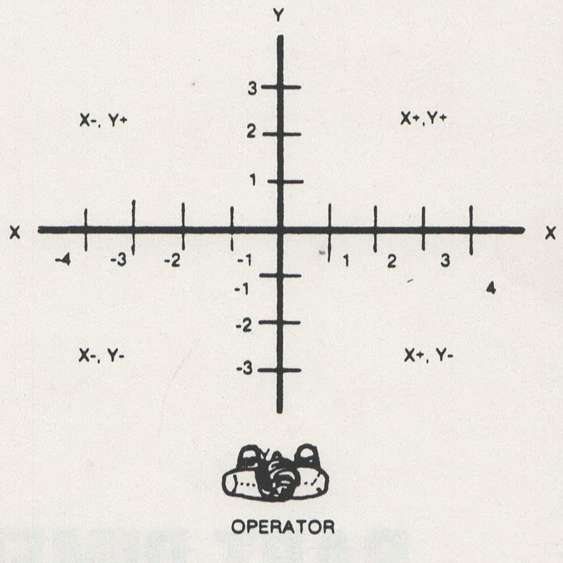


Figure 6.

Think of the X and Y axis both starting at absolute zero and extending outward along the axis to form quadrants. All dimensions to the right of zero are X+; to the left of zero, X-. All dimensions above zero are Y+, below zero, Y-.

To move the tool to a position that is to the right of zero and above zero, we must program an X+ dimension, and a Y+ dimension. The quadrants are shown with the plus and minus signs for all absolute dimensions which are in that quadrant. To move the tool to the left of zero and above it, we must program an X- dimension and a Y+ dimension.

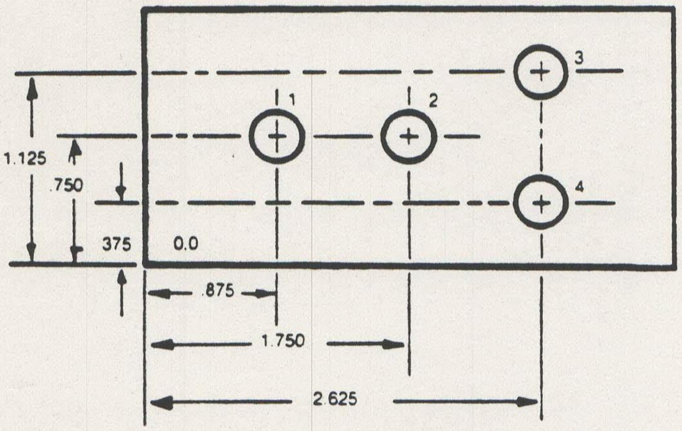


Figure 7

Figure 7 shows a plate with holes dimensioned from the lower left corner. Because it is dimensioned like this, absolute zero should be set at that corner and the tool motion will be referenced to that corner. If this corner of the part was placed on the quadrant diagram, the part would be in the upper right quadrant. This means all X and Y dimensions given to move to the hole locations will be X+ and Y+ dimensions because they are above and to the right of absolute zero. Write down the absolute dimension for holes 1, 2, 3, and 4. Check your answers with the answers below.

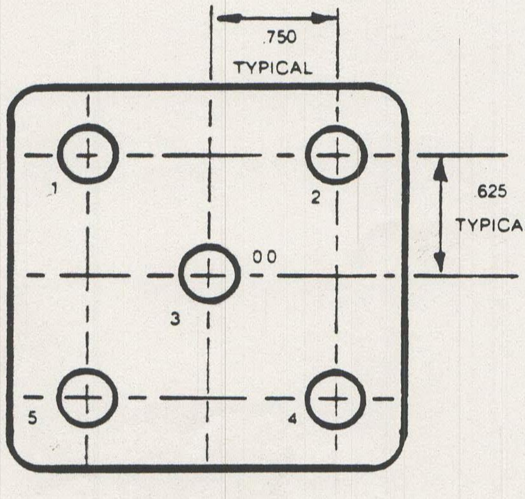


Figure 8

Figure 8 shows a part dimensioned from a center reference. The plus and minus signs of the hole locations will be determined by their position from absolute zero just as in figure 6. Write down the dimensions with the correct plus and minus signs for holes 1, 2, 3, 4 and 5 in figure 8. Plus signs can be omitted because they are understood by the control. If a dimension does not have a minus sign, it is assumed to be plus. Check your answers with the answers below:

Absolute dimensions for figure 7.
1. X .875 Y .750 ABS (ABSOLUTE)
2. X1.750 Y .750 ABS
3. X2.625 Y1.125 ABS
4. X2.625 Y .375 ABS

Absolute dimensions for figure 8.
1. X-.750 Y .625 ABS
2. X .750 Y .625 ABS
3. X 0 Y 0 ABS
4. X .750 Y-.625 ABS
5. X-.750 Y-.625 ABS

C. INCREMENTAL DIMENSIONS

Up to now we have used the absolute system of measurement to describe tool motion. Figures 9 and 10 show a need for a different type of measuring system. Notice that there is an absolute zero and that the first hole or slot is referenced to it but the remaining holes and slots are referenced from hole to hole or slot to slot. These are known as incremental dimensions. They are dimensioned not from absolute zero but from the previous position.

INCREMENTAL DIMENSIONS

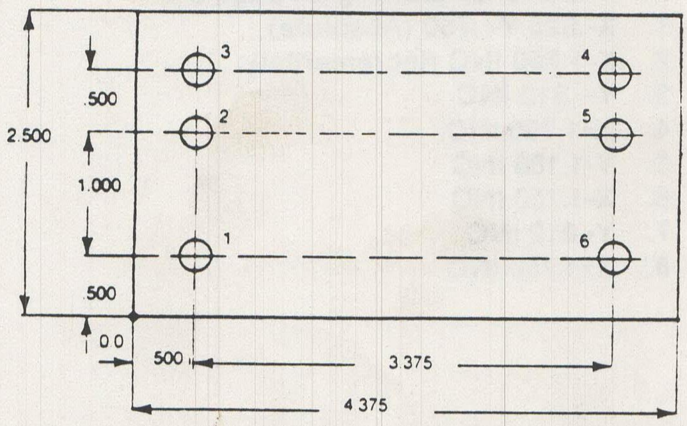


Figure 9

In the top example, hole location 1 is located at X+.500 Y+.500 from Absolute Zero. Hole location 2 is located Y+1.000 incrementally from hole location 1. Hole location 3 is located Y+.500 from hole 2, hole 4 is located X+3.375 from hole 3. From hole 4 we must move in the Y- (minus) direction to hole 5. The incremental distance from hole 4 to 5 is Y-.500. Similarly, hole 6 is Y-1.000 incrementally from hole 5. In these examples, incremental dimensioning and movement do not relate to absolute zero. Incremental dimensions are always given as a distance and tool direction. The sign of the dimension is the direction that tool is moving when using incremental. Figure 10 shows a series of slots. Again, the right center of the top slot is given in absolute. (position 1) Write down the absolute dimension of the first position and the incremental dimensions to move to the remaining positions. Check your answers with the answers below.

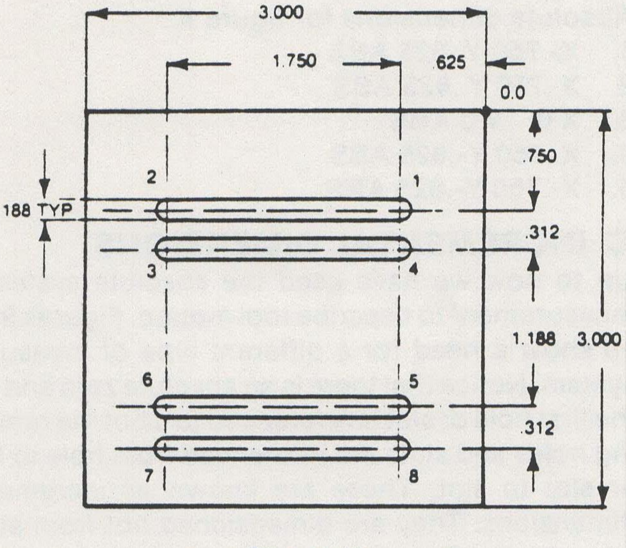


Figure 10

Dimensions for drawing on page 2-3:

1. X-.625 Y- .750 (Absolute)
2. X-1.750 INC (Incremental)
3. Y- .312 INC
4. X+1.750 INC
5. Y-1.188 INC
6. X-1.750 INC
7. Y-.312 INC
8. X+1.750 INC

SECTION 3: VARIOUS MODES OF OPERATION

All of the keys described in this section are keys that will determine the operational condition of the control. Each section will describe the purpose and use of each of these keys.

A. STARTING THE SERVO MOTORS

The servo motors will not start if one of the machine axis is at the end of travel and tripping a limit switch. Figure 11 shows the normal location for the limit switches and trips on the X and Y axis. Locate the switches on your machine and check to see that they are not being tripped.

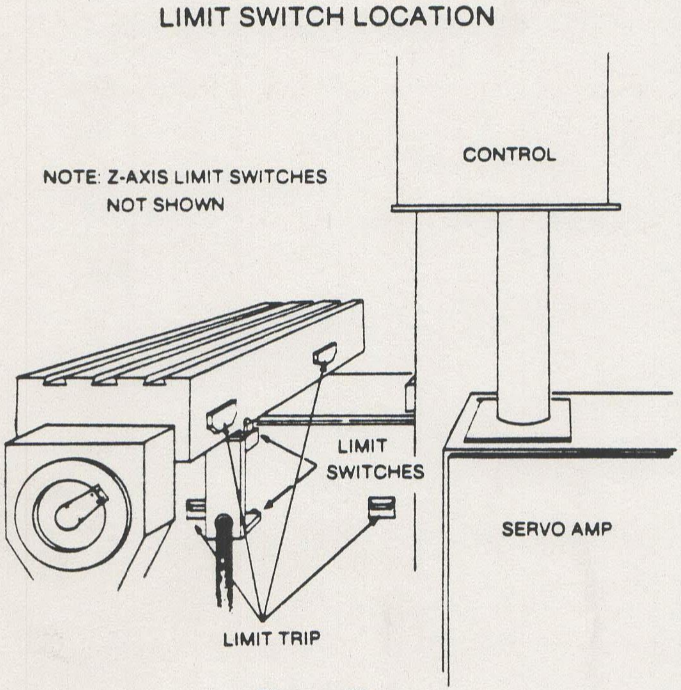


Figure 11

The limit switches on the Z axis are behind the cover and not visible when the cover is in place. The trips are set at the extreme top and bottom of the stroke. Move the quill with the handle or knurled knob to the center of its stroke to insure you are not tripping a Z axis limit switch.

To start the servo motors:

1. Push in then pull out the EMERGENCY STOP button on the console.
2. Put the MANUAL/SPINDLE/AUTO switch, located on the back of the servo amp cabinet, in the AUTO position.
3. Press the reset button located on the servo-amp cabinet.

If the console is ON, the servo motors will start. No movement will take place. To turn off the motors, press EMERGENCY STOP. This will not cause a loss of memory or position.

B. MANUAL

The MANUAL mode provides for manual positioning of the machine using the servo motors. Lower the knee to insure safe, unobstructed movement. If the servo motors are not on, turn them on using the instructions in section 3A. Now press the **MANUAL** mode button. The **MANUAL** button will light and the **RAPID** light just above it will come on.

The machine can now be moved in rapid traverse, which is 100 inches (2540mm) per minute (this value may be different depending on your particular machine and servo-motor combination). To move the machine, press the **X+** button. The machine will move at 100 IPM for as long as you hold the button or until a limit switch is reached. Again, notice that the **X+** represents tool motion.

If a limit switch is tripped, move off the trip with the hand-wheel and restart the servo motors.

The **IN POSITION** light will go out when the machine is moving, showing that the machine is not in position. The actual speed at which the machine is moving will always be shown in the feedrate display. By pressing the hand button (center) once, the light will change from **RAPID** to **FEED.** Feed is 40 inches (1000mm) per minute. The machine can be positioned the same way in feed using the axis buttons. Use the axis buttons to move the machine in feed.

Press the HAND button once again and the light will appear at JOG 3, which is an incremental move of .100 (1.00mm). JOG 2 is an incremental move of .010 (.10mm). JOG 1 is an incremental move of .001 (.01mm). When a JOG button is lit, the axis will move that increment each time an axis button is pressed.

C. SETTING THE ABSOLUTE ZERO

Zero can be set anywhere on the part. To set the X axis to zero:

1. Position the center of the spindle at the X. 0 reference point of the part.
2. Press MANUAL button (even if MANUAL is lit) to clear any previous command.
3. Press X, 0, EVENT ENTER.

At this time the X axis will be reset to zero and zero will be shown in the X axis display. Any axis can be set to zero using this method by pressing the correct axis button, 0, and EVENT ENTER. Reset each axis to zero, move the machine and notice the absolute dimensions in the displays will refer to this new zero.

D. PROGRAM ENTER

PROGRAM ENTER is the programming mode. To enter a program into memory, clear memory, add to or delete from a program, record a program on cassette tape or play a program into memory from cassette tape, you must be in the PROGRAM ENTER mode. All of these functions will be discussed later in detail. (See Section 5: Program Entry.)

E. PROGRAM CHECK

PROGRAM CHECK is used to verify what the operator has put into memory. Refer tc Section 5: Editing, for a practical use of the PROGRAM CHECK button.

F. SINGLE STEP

SINGLE STEP is used to execute the program one event or one move at a time. This can be used to dry run and check a program. If the START button is pressed in SINGLE STEP mode and AUX code 1900 is active (see Section 4.1) only 1 event at a time will be executed. If AUX code 1901 is active, then one X,Y move will be executed each time the START button is pressed in SINGLE STEP mode.

G. AUTOMATIC

Once the program has been stepped through and is known to be correct, the program can be run in automatic (AUTO). This is the normal operating mode for production.

H. FEED%

FEED % can be used to override a feedrate in MANUAL, AUTO or SINGLE STEP. If the feedrate being used is too fast, the feedrate can be reduced. This is done by pressing the button with the arrow pointing down in the FEED % section. Each time the button with the arrow pointing down is pressed, the feedrate will be decreased by 5%.

Press MANUAL, then the HAND button once. Move the machine in FEED with an axis button and at the same time press the button with the arrow pointing down in the FEED % section. Each time the button is pressed, the feedrate in the FEED display will decrease and the FEED % display will show the percentage of the feed being used.

Similarly, if the feedrate is too slow, it can be increased. To do this, press the button with the arrow pointing up in the FEED % section. This will increase the feedrate by 5%. The feedrate can be overridden from 0-120%. The FEED % override will effect all feedrates in the program.

Normally FEED % override does not effect rapid moves. There are times during set up or dry runs when it is desirable to use FEED % override for rapid moves. To accomplish this press MANUAL, AUX 1401, START. All rapid moves will then be overridden with FEED % override. To cancel FEED % override affecting rapid moves, press MANUAL, AUX 1400, START. Rapid moves will then be at 100 IPM.

I. INCH - MILLIMETER

This light indicates which type of measurement is being programmed and displayed. When using canned cycles, all variables must be entered in either inch or millimeter, not part in inch and part in millimeter. Under certain conditions, it is possible to accumulate a roundoff error when using circular motion in a Do Loop. If this condition should arise, enter and execute the program in millimeter.

SECTION 4:
PROGRAMMING KEYBOARD DEFINITIONS

This is a brief explanation of the buttons on the keyboard. They will be explained in detail in the example programs

A. DISP POSN (DISPLAY POSITION)

When the **display position** button is lit, the readouts will be displaying the absolute position of X, Y and Z. This will always be the absolute position no matter what mode you are in. Pressing **display position** will turn it off and the readout will display the current programmed move in all modes except **MANUAL.** In *MANUAL* with **display position** off the readouts will be blank because there is no programmed move. **Display position** can be turned on or off at any time to see absolute position or the programmed move.

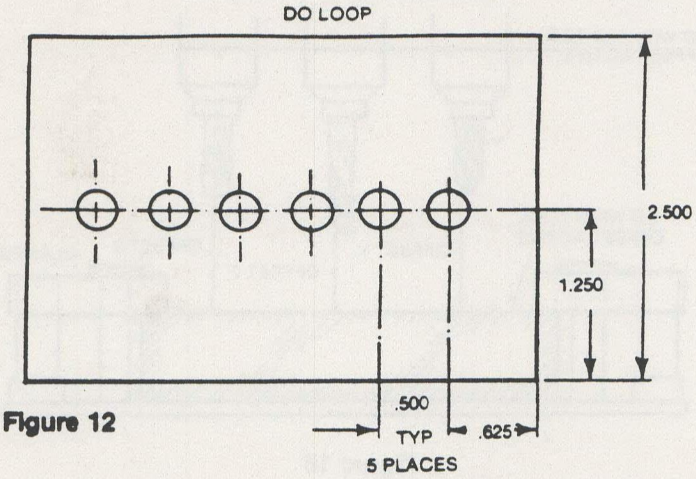


Figure 12

B. DO (DO LOOPS)

Also referred to as "step" and "repeat", a do loop can be used for anything that is repeated over an equally spaced distance. Figure 12 shows a row of holes that are equally spaced at .500 along the X axis. A do loop is used to eliminate the programming of each hole. **DO** is entered with a number (**DO 5**). This is the number of times the programmed operation is repeated.

Figure 13 shows a series of windows or pockets which again are equally spaced. These represent typical examples of when a Do Loop can be used. See example 2 for using Do Loops.

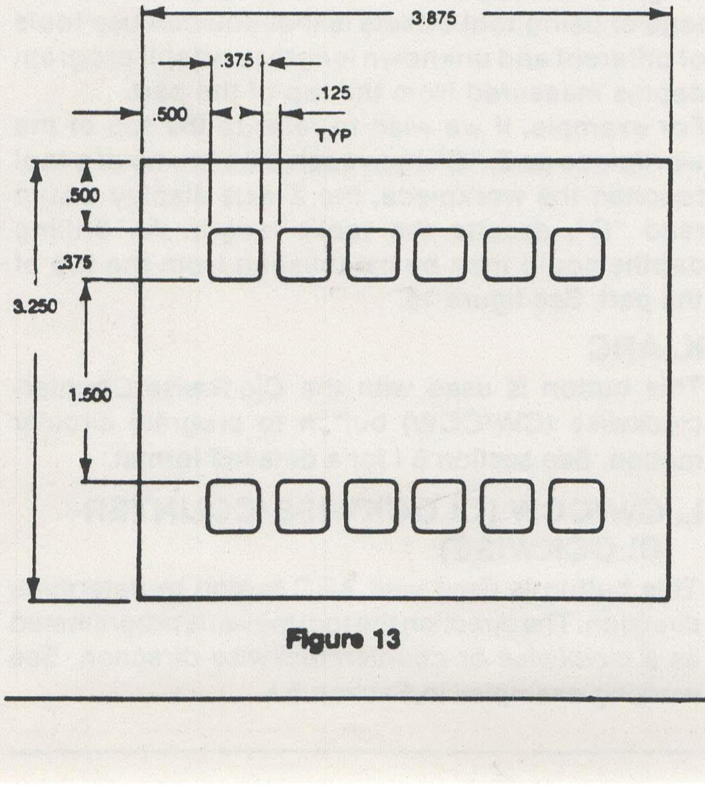


Figure 13

C. SUBR (SUBROUTINE)

Subroutines are also referred to as mini-programs. Subroutines can be used for any section of the program which has to be repeated. In figures 14 and 15, subroutines could be used to repeat the pattern of holes or pockets. A subroutine is always stored in memory after the end of the main program and can be recalled at any time for executing. See example 3, 4, 6 and 7 for subroutine examples.

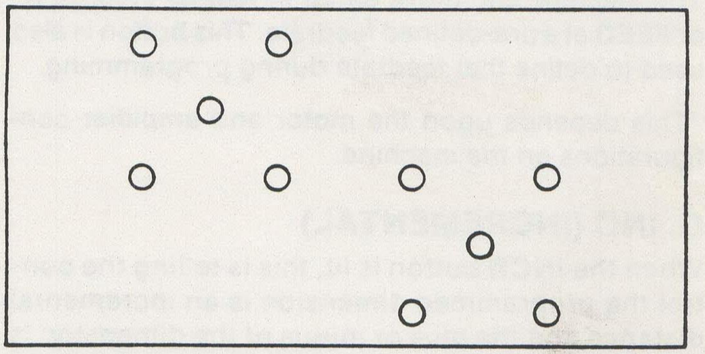


Figure 14

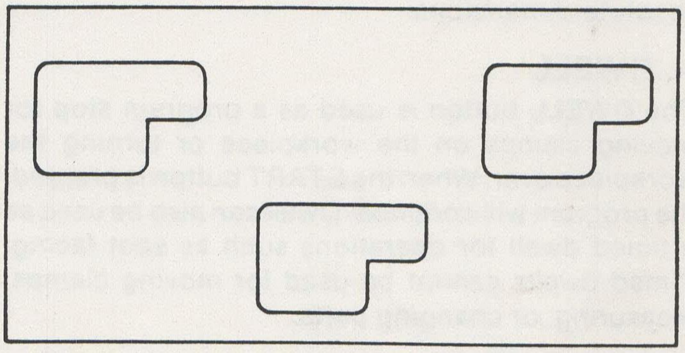


Figure 15

D. CALL

The **CALL** button is used in the main program to execute a subroutine. The subroutine is assigned a number (i.e. SUBR 1) and then is called (i.e. CALL 1) to execute the subroutine. Subroutines can be labeled from 1 to 9000.

E. END

There are three uses for the **END** button.

1. In a do loop, **END** is programmed after the last event to be repeated. This stops the do loop but the program will not stop.

2. Because a subroutine is a "mini-program", it must have its own **END** statement. This ends the subroutine but the program does not stop.

3. The **END** button is used at the end of the main program to stop motion. When the end of the program is reached, by pressing the START button, the program will return to event 1.

F. RAPID/FEED

This button is used in programming to specify the speed at which the control will move the machine. The machine can move either in **RAPID** (100/ipm)* or **FEED** at a pre-defined feedrate. This button is also used to define that feedrate during programming.

*This depends upon the motor and amplifier configurations on the machine.

G. INC (INCREMENTAL)

When the **INCR** button is lit, this is telling the control the programmed dimension is an incremental distance and the plus or minus of the dimension is the tool direction. Since dimensions can only be entered in incremental or absolute, when the **INCR** button is off, the dimensions in the displays are absolute dimensions.

H. DWELL

The **DWELL** button is used as a program stop for moving clamps on the workpiece or turning the workpiece over. When the START button is pressed, the program will continue. Dwell can also be used as a timed dwell for operations such as spot facing. Timed dwells cannot be used for moving clamps, measuring, or changing parts.

??

I. AUX (AUXILIARY)

The **AUX** button is used in the main program to cause variations in the standard control functions. These variations are assigned numbers and entered into the program prefixed by the **AUX** button (i.e. **AUX 7, AUX 400, AUX 2500, etc.). AUX** codes control such features as Mirror Image, Software Limits, Absolute Zero Shift etc. Included in the Miscellaneous section of this manual is a complete listing of all the **AUX** codes and their functions in both the standard Crusader control and the optional Advanced Software System - 1 package.

J. TOOL (TOOL OFFSETS)

To use this feature, you must have a quick change tooling set so when a tool is replaced in the spindle it will always maintain the same length. See Section VI-F for detailed instructions for setting Tool Offsets.
Study figure 16

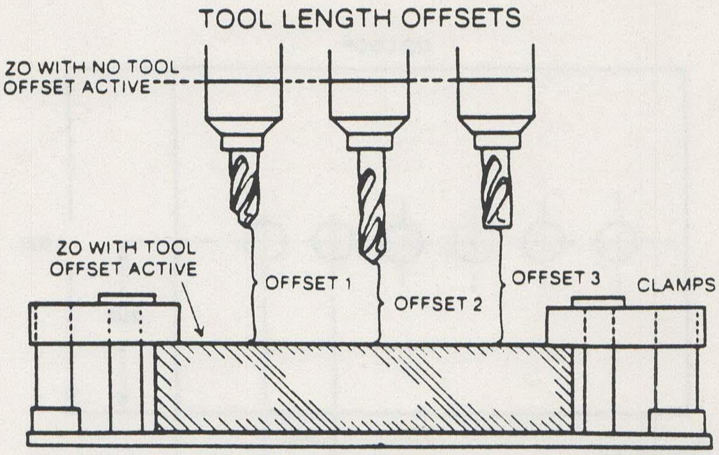


Figure 16

Tool offsets, also referred to as tool length compensation and T.L.O., are used to compensate for various lengths of different tools.
The distance between the tip of the tool with the quill fully retracted and the top of the part is the length of compensation for that tool. The advantage of using tool offsets is that you can use tools of different and unknown lengths and still program depths measured from the top of the part.
For example, if we wish to refer to the top of the workpiece as Z, "O" then each time the tip of a tool touched the workpiece, the Z-axis display would read "O", despite the tool's length. All drilling depths could then be established from the top of the part. See figure 16.

K. ARC

This button is used with the Clockwise/Counterclockwise (CW/CCW) button to program circular motion. See section 5 I for a detailed format.

L. CW/CCW (CLOCKWISE/COUNTER-CLOCKWISE)

This button is used with ARC button to determine direction. The direction the tool moves is programmed as a clockwise or counterclockwise direction. See working examples in Section 5 I.

M. G (G CODES)

See figures 17-23

G80 SERIES CODE

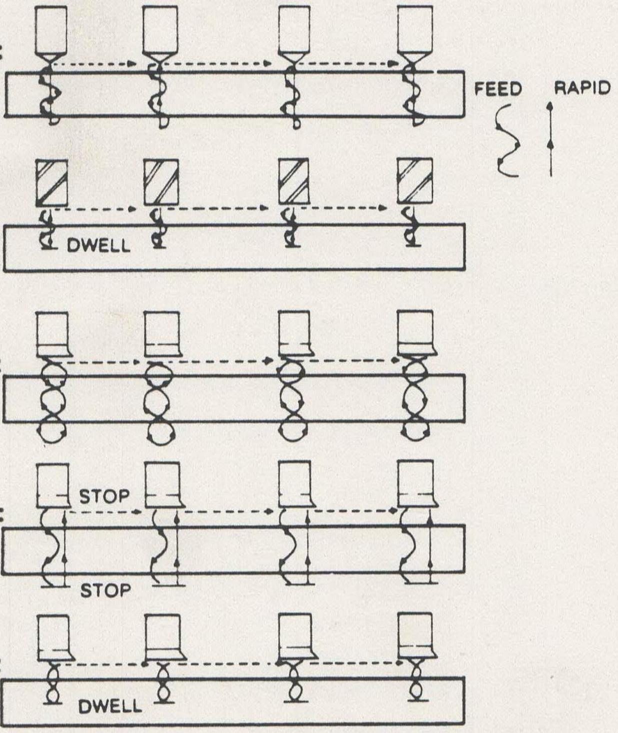


Figure 17: G-81

Figure 18: G-82

Figure 19: G-85

Figure 20: G-86

Figure 21: G-89

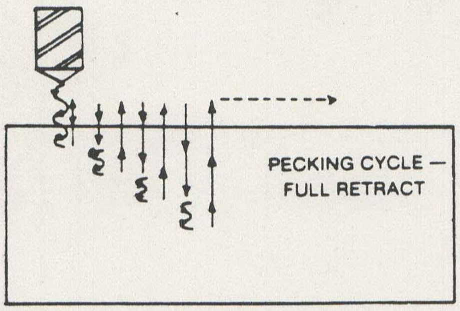


Figure 22: G-83

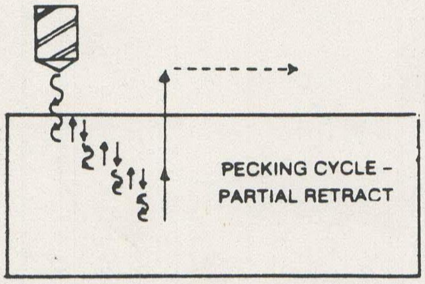


Figure 23: G-87

The G 80 series are canned drilling cycles. Each G code simulates the Z-axis motion done by an operator for hole drilling or boring. A brief Description of each G-code function is listed below:

G 80: Deactivates all drilling canned cycles.
G 81: (Basic Drilling Cycle) Feeds in to depth, rapids out.
G 82: (Counter-boring or Spot-facing Cycle) Feeds in to depth, dwells for programmed time, then rapids out.
G 83: (Peck Drilling Cycle) Feed in, rapids out, rapids in, feeds in, rapids out, rapid return until final depth is reached, rapids out.
G 85: (Boring Cycle) Feeds in to depth and feeds out.
G 86: (Boring in One Direction) Feeds in to depth, program stop, rapids out, program stop.
G 87: (Chip Breaking Cycle) Feeds in, retracts .050, feeds in, retracts .050 until final depth is reached, rapids out.
G 89: (Flat Bottom Boring Cycle) Feeds in to depth, dwells for a programmed time, then feeds out.

All G-80 series codes stay active until deactivated by G 80. Using EVENT SEARCH or changing modes (except changing from SINGLE STEP to AUTO) will also deactivate G codes. See examples 2-5 for G-80 examples.

The G-codes for cutter compensation follow:

G 40: Deactivates cutter diameter compensation (see section 9)
G 41: Activates cutter diameter compensation to the left.
G 42: Activates cutter diameter compensation to the right.

See section 11 at the back of this manual for G series codes used with the optional Advanced Software System-1 package.

N. VARIABLE BUTTON (V)

This is used to assign a value (such as feedrate, hole depth, pocket length, etc.) when using G-codes.

V 20: Specifies feedrate for Z-axis moves in Drilling Canned Cycles.
V21: Specifies the height above the workpiece at which the tool begins and ends after each hole is machined. This height **must** be absolute.

NOTE:

When using peck cycles, always use a starting height plane of .100 above the actual work surface. (2 mm if programming in metric)*

V 22: Specifies dwell time when using G 82 or G 89.

V 23: Specifies the maximum amount of each peck in Pecking cycles. This distance must be given as a positive incremental dimension. (The control will adjust this dimension to ensure that each peck is of equal distance, but will not exceed the maximum amount of each that has been specified by the operator.)

See section 11 at the back of this manual for V-codes used with the optional Advanced Software System-1 package.

*This is a default distance. To alter this distance, see section 7.

3 ?

SECTION 5:
PREPARATION FOR PROGRAMMING

A. HOLDING THE WORKPIECE (STEP 1)

Machining practices that were learned and have become second nature now must be put in the program. The Crusader cannot operate efficiently without the programmer using good machining practices as the finished parts will still reflect the machinist's skill. The machinist must still find the most rigid and best way of holding the workpiece.

The writing of the program will depend on how the workpiece is being held. If clamps will be used, the tool must be programmed to move around them. Any obstructions will have to be kept in mind while programming to avoid collisions. How the workpiece is held will determine which part dimensions are along the X axis and which dimensions are along the Y axis. For these reasons, the way the workpiece will be held must be decided before writing the program.

DETERMINING A REFERENCE POINT

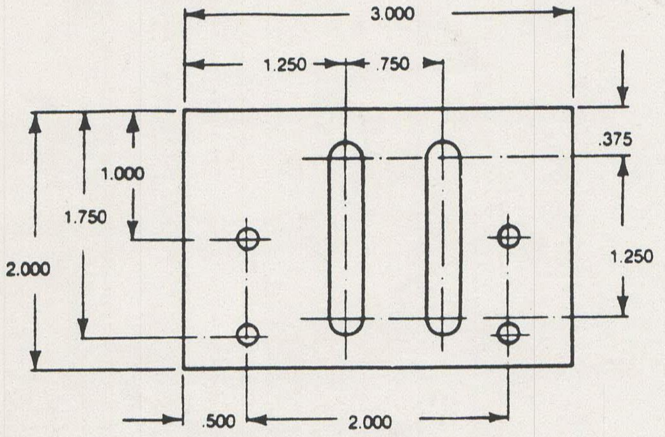


Figure 24.

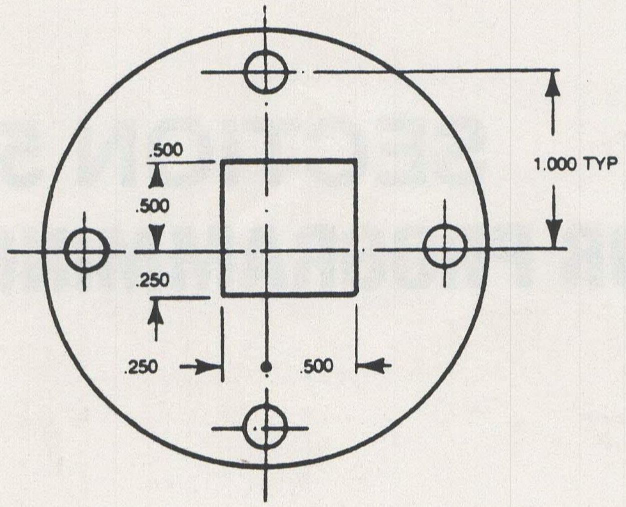


Figure 25

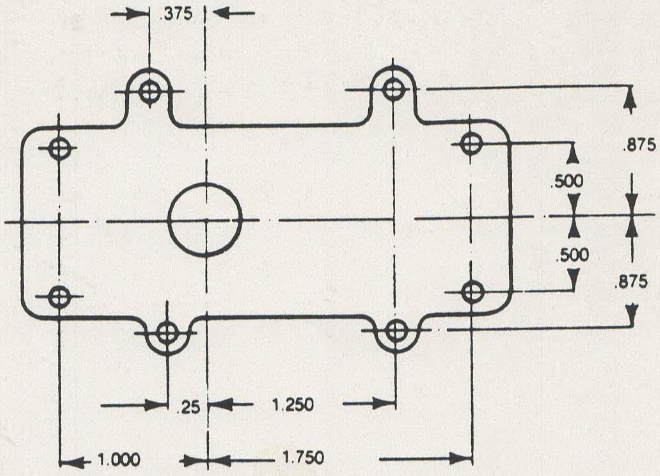


Figure 26

B. DETERMINING A REFERENCE POINT (STEP 2)

The next step is to pick a point of reference (a place to set X and Y absolute zero). This will usually be the edge of the part (from which part dimensions are given), or the center of a bore or hub on a round part. With a Crusader, the absolute zero can be set at any point. It is best to use the same point used on the print. Sometimes a pin or hole in a fixture is used as an absolute zero if this is what locates the part.

Study the three examples (figures 24-26). Figure 24 should have X and Y zero set at the top left corner, Figure 25 should have X and Y zero set at the center. Figure 26 should have the X and Y zero set at the large reference hole.

The way the part is held may make a difference in where the absolute zero is set. For this reason you should decide how to hold the part and where to set absolute zero before trying to program the part.

C. TOOL SELECTION (STEP 3)

Examine the blueprint to determine what tools will be used to machine the part within the tolerances given. Next, determine the order in which the tools will be used. For example, by looking at Figure 27 we can determine that:

1. The workpiece will be held in a vise with a workpiece stop on the left side of the part.
2. Absolute zero will be set at the upper left corner because most dimensions reference that corner.
3. Tool 1 will be a spotting drill; Tool 2 a .280 drill; Tool 3 a .406 end mill for counterboring (it will also be used to mill the .751 slot).

The chip loads or speeds and feed of the cutting tools are programmed in inches (or millimeters) per minute and must be within the horsepower range of the spindle. This information can be calculated by using speed and feed calculators or one of the many machining handbooks available. Tooling speeds and feeds used in the examples in this manual are theoretical but represent a practical range for the tooling used.

After the tooling is selected, a logical sequence must be determined and followed. For example, you would not drill a hole before center drilling, so number the tools in the order they will be used.

TOOL SELECTION

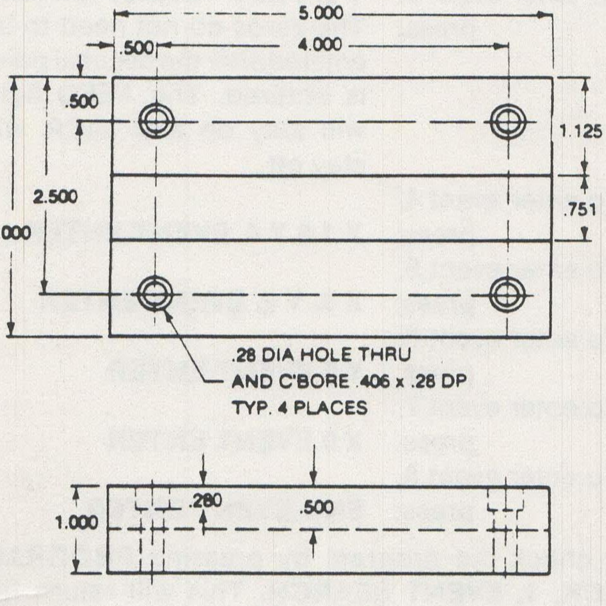


Figure 27

D. BEGINNING A PROGRAM (ONE-AND TWO-AXIS LINEAR INTERPOLATION)

In this section you will write a program, enter it in the control and execute it to "dry run" the machine to points 1 thru 5 as drawn in figure 28. You will program the X and Y axis only and use this simple exercise to become familiar with entering information. Absolute zero will be set at the lower left corner and the spindle will be programmed to move to points 1 thru 5 at 20 ipm.

As the machine moves to points 1-5; notice that you have caused the control to execute three basic moves with the machine: an X-axis linear move, a two-axis linear move (angle) and a Y-axis linear move.

In order to keep the program in order and in understandable form, it is best to write it all down. The programming sheets provided with this manual will offer you a logical, concise, step-by-step fromat to follow. The sheets can be filed away for future reprogramming should the part ever return to your shop. Only file sheets that contain programs tested and proved in dry runs.

PROGRAMMING EXAMPLE

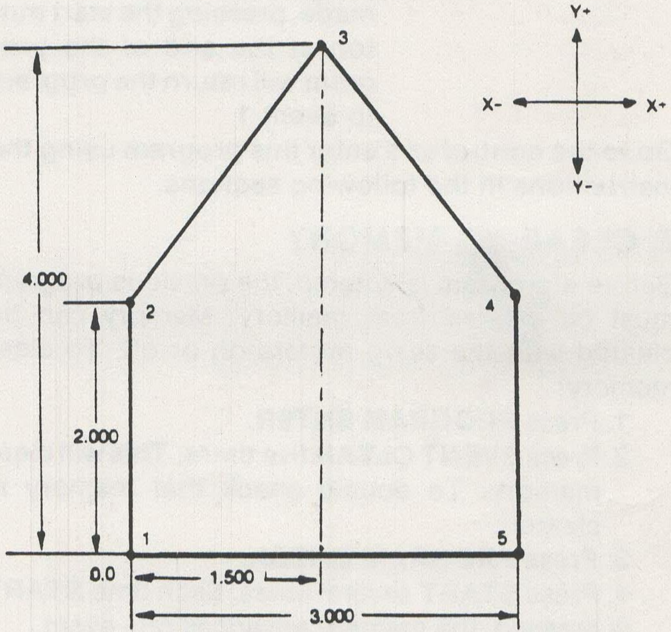


Figure 28

The steps in the program are numbered starting at 1 and are called "events". Up to 999 events can be stored in memory. This program will have the following events:

	EVENT	FUNCTION
1.	FEED 20.0	This sets the feedrate at 20 ipm.
2.	X O Y O FEED ABS	This moves the spindle to point 1 (All dimensions are absolute unless otherwise noted
3.	Y 2.000 FEED ABS	Moves the spindle to point 2. Since no X axis motion is programmed, the X axis does not move. (Single-axis linear move).
4.	X 1.500 Y 4.000 FEED ABS	Moves the spindle to point 3 (Two axis linear move for angle milling)
5.	X 3.000 Y 2.000 FEED ABS	Moves the spindle to point 4

6. Y O FEED ABS	Moves the tool to point 5. Again no X axis motion takes place.
7. X O FEED ABS	Moves the tool to point 1. No Y axis motion takes places. (Single axis linear move).
8. END	This ends the program. If more that one part is to be made, pressing the start button at the end of the program will return the program to Event 1

Go to the control and enter this program using the instructions in the following sections.

E. CLEARING MEMORY

Before a program is entered, the previous program must be cleared from memory. Memory can be cleared with the servo motors on or off. To clear memory:

1. Press **PROGRAM ENTER.**
2. Press **EVENT CLEAR** five times. This will clear memory. To double check that memory is cleared:
3. Press **PROGRAM CHECK.**
4. Press **START** several times. Each time START is pressed, the memory advances one event.

There should be nothing showing in the displays. This indicates the events are empty and memory has been cleared. The event number is shown in the upper left display. We are now ready to enter the program shown in section D (Beginning a Program).

F. PROGRAM ENTRY

After memory has been cleared and checked, the program can be entered. First press **PROGRAM ENTER, 1, EVENT SEARCH.** This will return the control to event 1 (which will be shown in the middle of the top display on the keyboard).

NOTE:
All words in bold-type represent actual keystrikes.

1. To enter event 1, press:	**FEED** (on the rapid/feed button), **20. EVENT ENTER.** This will store event 1 in memory and advance to event 2. Every event must entered. If the wrong information is pressed before pressing **EVENT ENTER,** press **EVENT CLEAR.** This will clear the event so the correct information can now be put in and entered.

NOTE
The dimensions listed in this manual make use of the decimal point key. Therefore, when "2." is seen it must be entered as "2" and "." instead of "2,0,0,0,0".

2. To enter event 2, press:	**X O Y O** (the **FEED** button will be lit to show the move is to be made in feed. If the INCR button is on, then turn it off. Press **EVENT ENTER**
3. To enter event 3, press:	**Y 2. FEED EVENT ENTER.** The zeros do not need to be entered after the decimal point is entered. The FEED light will stay on and INCR will stay off.
4. To enter event 4, press:	**X 1.5 Y 4. EVENT ENTER**
5. To enter event 5, press:	**X 3. Y 2. EVENT ENTER**
6. To enter event 6, press:	**Y 0 EVENT ENTER**
7. To enter event 7, press:	**X 0 EVENT ENTER**
8 To enter event 8, press:	**END EVENT ENTER**

Now check the program by pressing **PROGRAM CHECK, 1, EVENT SEARCH.** This will return the program to event 1. If the information is correct in the displays, press START to advance to event 2. If the information is correct, press start each time to advance through the program. If an error is found, see the editing instructions in the next section.

G. EDITING

If an error is found in an event being displayed press **PROGRAM ENTER,** the event number of the event to be edited, then **EVENT SEARCH.** Press **EVENT CLEAR** then press the buttons to display the correct information. Press **EVENT ENTER** and the correct information will be stored in memory. To continue checking the program, press **PROGRAM CHECK.** Continue checking the program by pressing **START** to advance through each event.

EVENT SEARCH can be used in any mode. Anytime you want to locate a certain event, press **EVENT CLEAR,** the event number, then press **EVENT SEARCH.**

When entering information, look at the displays before pressing **EVENT ENTER**. Check to see that everything which is supposed to be entered in the event is displayed. Also check to see that nothing is displayed which should not be. By looking at the displays you can see exactly what is being entered. If this is done you will nearly always enter the correct information.

For more information on program editing, refer to Example 1, Section 6-C. This section will explain the use of the **ADD-DELETE EVENT** buttons in a practical example.

H. PROGRAM EXECUTION

Lower the knee to insure safe, unobstructed movement. After the program has been entered and checked, use the manual Jog buttons and set the absolute zero in the center of the machine travels so the axis can move far enough to execute the program. Turn on the servo motors. Press **SINGLE STEP**, 1, **EVENT SEARCH**. This will return the control to Event 1.

Press START. The control will read Event 1 which sets the feed rate. Press start again and the control will execute Event 2. If the machine is not at absolute zero, it will feed to absolute zero. Each time the **START** button is pressed, one event will be executed. Press **AUTO** then **START** to do the program in auto. If a limit switch is tripped, move the absolute zero to allow the machine to execute the program

The following chapters will outline several working examples. These examples will "walk you through" most of the Crusader's features and will familiarize you with its capabilities.

I. CIRCULAR MOTION

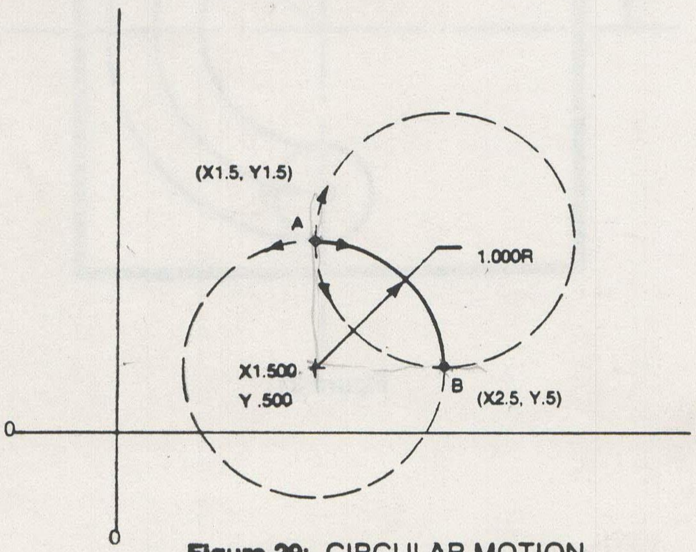


Figure 29: CIRCULAR MOTION

Figure 29 shows an example of circular motion from point A to point B taking several paths. The path we are to program is the solid line.

Let's examine the arc we want to program and determine what makes it unique from the other arcs which go from point A to point B. The start point (A) and finish point (B) are common to each arc. If this is all that was programmed for the arc, the Crusader would not know which path to take. The direction of the arc we want to program is clockwise.

If the start point, direction clockwise, and end point were programmed, it still would leave two possible paths. By describing the X and Y coordinates of the arc center (which also determines the radius) the last incorrect path is eliminated. It is because we must eliminate all incorrect paths that the following rules are used.

All arcs must follow this format. Each rule represents one event in the program.

1. Position the tool to the start point of the arc (X and Y.
2. Position the tool into the workpiece if it is not at the correct depth (Z).
3. Arc and direction (CW or CCW).
4. Center point of arc (X and Y).
5. Finish point of arc (X and Y).
6. Arc.

NOTE:
If a feedrate is not entered for the arc, the control will use the feedrate that was last entered into the program. If no feedrate was entered previously. The control will assume a feedrate of 40 inches per minute

When the center point or finish point is described as an incremental dimension, the distance is given from the start point. For a full circle (360 degr.), omit the 5th step (finish point) and the control will assume the start point is the finish point.

Use the rules stated above as guidelines and follow the program written below to see how each rule is followed.

EVENT
1. X 1.500 Y 1.500 R (RAPID) A
2. FEED 10.0
3. ARC CW
4. X 1.500 Y .500 (Feed) A ℄ CIRCLE
5. X 2.500 Y .500 F A
6. ARC
7. END

Enter and execute this example.

Use Figures 30 and 31, and following the format given, do a full circle in absolute using a .500 end mill. Enter and execute each example below (note: no Z axis move is programmed as this is for a demonstration only). Remember, you are programming the center of the tool:

EVENT

(1) **Full Circle Absolute**

1. X3. Y-1.25 F A	Position to start point
2. Arc CW	Arc and direction
3. X3. Y-3. FA	Absolute position of center of arc.
4. Arc	
5. End	

(2) **Full circle incremental (same part):**

1. X3. Y-1.25 F A	Absolute position of the start point
2. Arc CW	Arc direction
3. XO Y-1.75 F I	The distance to the center point of the arc given from the start point of the arc.
4. Arc	Arc Finish Command
5. End	End of Program

(3) **Format for Absolute half circle:**

1. X3. Y-1.25 F A	Position of start point
2. Arc CW	Arc direction
3. X3. Y-3. F A	Center point
4. X3. Y-4.75 F A	Absolute position of end point
5. Arc	Arc Finish Command
6. End	End of Program

(4) **Half circle incremental:**

1. X3. Y-1.25 F A	Position of start point
2. Arc CW	Arc direction
3. XO Y-1.75 F I	Distance to center of arc given from start point of arc.
4. XO Y-3.5 F I	Distance to finish point given from start point of arc.
5. Arc	Arc Finish Command
6. End	End of Program

The maximum feedrates possible for extremely small radii are listed below:

Radius	Max. Feed
.010 R	50 IPM
.005 R	25 IPM
.002 R	15 IPM
.001 R	10 IPM

If arcs are programmed with less that a .001 radius, a linear move will be made from the start point to the end point.

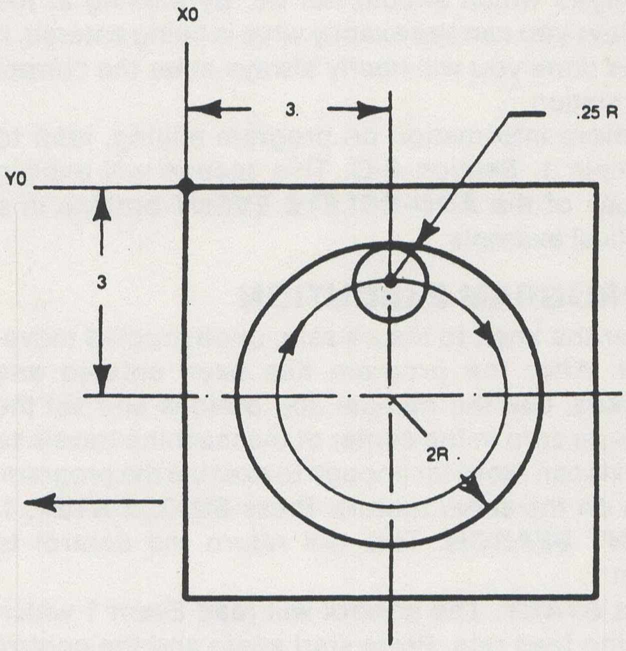


Figure 30

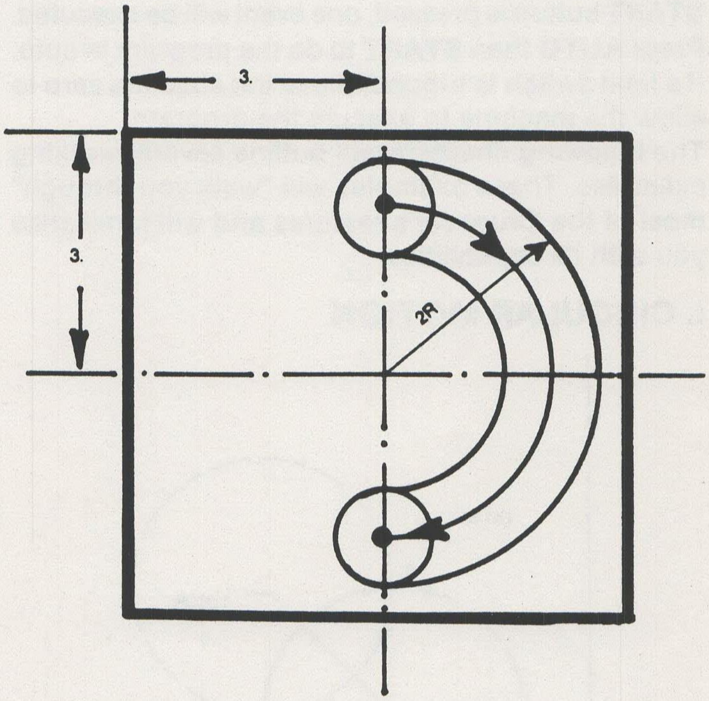


Figure 31

SECTION 6:
PROGRAMMING EXAMPLE (DRILLING AND SPOT DRILLING)

A. EXPLANATION OF EVENTS (EXAMPLE 1)

The best way to use this example is to actually machine a blank like the one shown in figure 35 and use this program to spot drill and drill the holes

After studying Example 1, it has been decided that the part will be held in a vise with a stop on the left side. The reference point on the print is shown in the upper left hand corner; set absolute zero in the same place. The part will be on parallels not more that 5/16 thick.

From the blueprint it has been determined that Tool#1 will be a 90 degree spot drill to center the holes and Tool #2 will be a .250 DIA drill.

Tool #1 will run at 3000 rpm and 6 ipm. Tool #2 will run at 2000 rpm and 4 ipm.

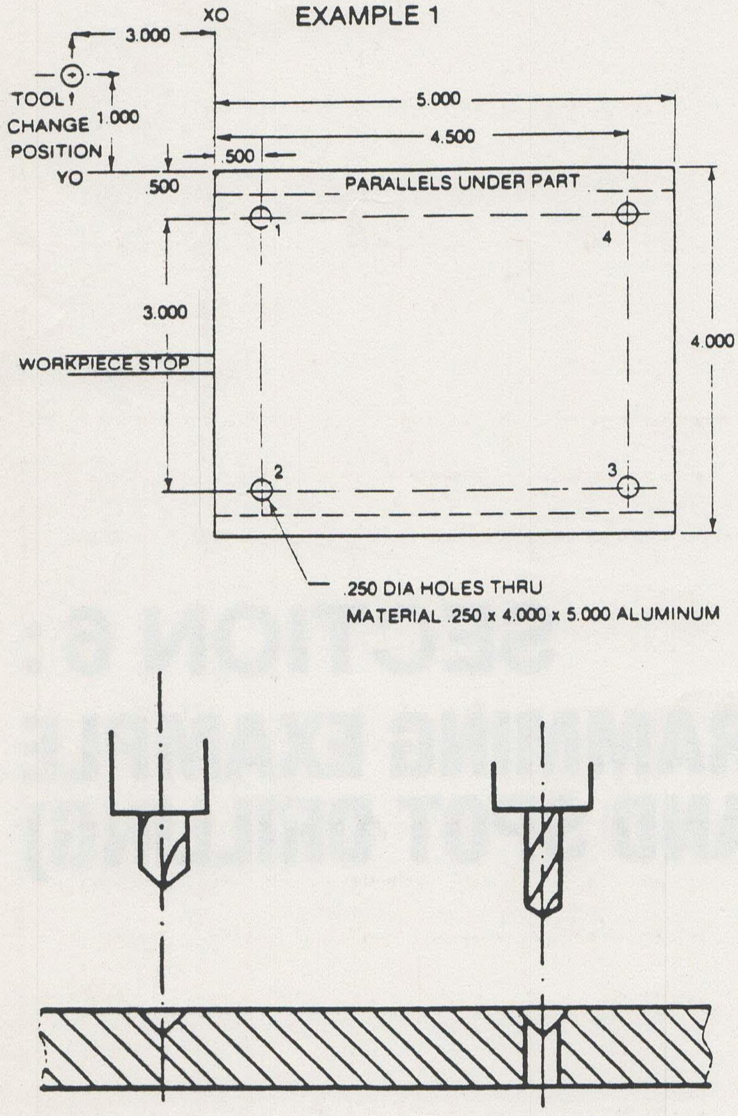


Figure 32

This information along with the program should always be written down before entering it into the Crusader. A sample programming sheet (on the next page) will be used to keep track of this information. While reading this section, use figure 32 (Example 1) for reference as you program the part.

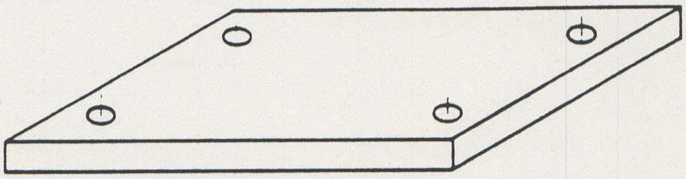


Figure 33: FINISHED PART

The machining sequence in the program will be to first list the tool offsets. Then the quill will be retracted and moved to X-3.000 Y 1.000. This is to move the spindle so the part can be loaded in the vise and Tool 1 can be put in the spindle. The program will stop and wait for the operator to do this. The holes will then be spot drilled. The quill will retract and move back to the tool change position. The operator will change tools and the holes will then be drilled. The quill will then retract and will again be moved to the tool change position. The program will end.

If a quick change tooling system is not available, a drill chuck can be used and the spot drill and drill can be inserted to the dead stop of the drill chuck to use as a repeatable stop.

The first sections of the programming sheet have already been filled in with the set up instructions and tool list that has been established.

The steps of the program will be numbered sequentially and called "events". The event numbers are located in the first column of the program sheet, the rest of the columns show the possible entries that can be made for each event.

It is important to establish a programming format that keeps safety in mind.

The first thing to do is create a table in the program to list the offsets of the tool to be used. The values for tool length offsets cannot be determined by the programmer when the program is written because the lengths of the tools are unknown. Offsets are to be entered by the operator at the machine during set up.

Event 1 of the program shows "Tool 1001". The "01" is the tool number. The prefix "10" is telling the control you are setting that tool's offset. Event #2 is left open for entering the value of that offset by the operator.

SHEET 1 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 1 PROGRAMMER JTM DATE 4.1.83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
TOOL 1: SPOT DRILL @ 3,000 RPMS TOOL 2: .250 DRILL @ 2,000 RPMS		WORKPIECE HELD IN VISE WITH X, Y ABS. "0" IN UPPER LEFT CORNER OF WORKPIECE. STOP IS ON THE LEFT SIDE OF PART. TOOL OFFSETS ARE SET AT THE TOP OF THE PART.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2												
3									1002			
4												
5									0			
6			0	R	A							
7	-3.	1.		R	A							
8									1			
9								V20: 6.				
10								V21: .1				
11							81					
12	.500	-.500	-.130	R	A							
13		-3.		R	I							
14.	4.500			R	A							
15		3.		R	I							
16							80					
17									0			
18			0	R	A							
19	-3.	1.		R	A							
20									2			

SHEET 2 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 1 PROGRAMMER JTM DATE 4.1.83

TOOL LIST OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.	SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
21								V20: 4.				
22								V21: .1				
23							81					
24	.500	-.500	-.35	R	A							
25		-3.		R	I							
26	4.500			R	A							
27		3.		R	I							
28							80					
29									0			
30			0	R	A							
31	-2.	1.		R	A							
32												END
3												
4.												
5												
6												
7												
8												
9												
0												

Anilam Electronics Corporation 5625 N.W. 79th Avenue, Miami, Florida 33166 / (305) 592-2727 / Telex 26-4024

Event #3 shows Tool 1002. Again, "02" represents the tool number. The prefix "10" is again telling the control that you are setting the offset for tool #2. Event #4 is left open for entering the value of that offset by the operator.

Event #5 shows Tool 0. This tells the control that no tool offset is active at this time. This is done to deactivate any offset. This is to be done at the beginning of every program.

Event #6 shows Z O R A. This is the first block that causes machine tool motion. Z O retracts the quill to the top of its travel. This is done as a safety measure to clear the part. Rapid (R) was chosen to move the quill at 100 IPM and Absolute (A) was chosen because Z O is an absolute position. (Refer back to section 4-J if events 5 or 6 are not clear.)

Event #7 shows X -3. Y 1. R A. This is a position off the workpiece where the operator can change tools. This could be any position that is convenient. In this case, the spindle will move to a position of X -3. Y 1. in rapid (100 IPM). This is an absolute position.

Event #8 shows Tool 1. This activates the offset for tool #1. This event will stop motion of the machine and allow the operator to change tools.

Event #9 shows V20: 6. This is the feedrate for drilling.

Event #10 shows V21: .100. This is the distance above the top of the part where the tool will start feeding into the workpiece. It also is the position the tool will retract to after the hole is drilled.

Event #11 shows G81. This will activate a canned cycle that feeds in and rapids out to spot-drill the holes.

Event #12 shows X .500 Y .500 Z -.130 R, A. This is the position and depth of the first hole. Because G81 is active, the tool will rapid to the X, Y location, rapid to the V21 dimension in Z (.100 above the workpiece), then feed at the V20 feedrate (6 IPM) to the Z axis depth, and finally retract to the V21 position. G81 will stay active until cancelled by G80.

Event #13 shows Y -3. R I. This is an incremental rapid move to hole 2. Since G81 is active, the spot-drilling operation will be repeated and hole 2 will be spot-drilled automatically.

Event #14 shows X 4.500, R, A. This is an absolute rapid move to hole 3. The spot-drilling cycle will again be repeated.

Event #15 shows Y 3. R I. This is an incremental rapid move to hole 4. The spot-drilling cycle will again be repeated.

Event #16 shows G80. This deactivates the drilling cycle. This must be done after the last hole is drilled.

Event #17 shows Tool O. This will deactivate tool 1's offset.

Event #18 shows Z O R A. This will fully retract the quill to the top of its travel.

Event #19 shows X -3. Y 1. R A. This will rapid the tool clear of the workpiece for a tool change.

Event #20 shows Tool 2. This will activate the offset for "tool 2". The machine will stop and allow the operator to change tools. See section 4-J for further explanation if necessary.

Event #21 shows V20: 4. This is the feedrate for drilling.

Event #22 shows V21: .100. This is the distance above the workpiece at which the tool will start feeding into the part. The tool will also retract to this point after each hole is drilled.

Event #23 shows G81. This activated a canned cycle to feed in, rapid out, to finish drilling the holes.

Event #24 shows X .500, Y -.500 Z -.350 R A. This will rapid the tool to the X, Y location of the first hole. Because G81 is active, the tool will rapid to .100 (V21) above the part, feed at 4 IPM (V20) to Z -.350 and rapid back to .100 above the part.

Event #25 shows Y -3. R I. This is an incremental rapid move to hole 2. Because G81 is active, the second hole will be drilled as described.

Event #26 shows X 4.500 R A. This is an absolute rapid move to hole 3. The drilling cycle will again be automatically repeated.

Event #27 shows Y 3. R I. This is an incremental rapid move to hole 4. The drilling cycle will again be repeated.

Event #28 shows G80. This deactivates the drilling cycle to stop the feed-in, rapid-out motion.

Event #29 shows Tool O. This will deactivat tool 2's offset.

Event #30 shows Z O R A. This will fully retract the quill to the top of its travel to clear the workpiece.

Event #31 shows X -3. Y 1. R A. This will rapid the spindle to the tool change position.

Event #32 shows END. This finishes the program. The machine will stop when this event is reached.

Go to the machine and clear memory and use the step-by-step instructions below and enter the program.

B. ENTERING THE PROGRAM (EXAMPLE 1)

First clear the memory by pressing **PROGRAM**

ENTER, then press **EVENT CLEAR** five times. This will clear the memory and return the control to event 1.

The program is to be entered exactly as on the program sheet. Remove the program sheets and use them to follow this sequence. Since this program is dimensioned in inches, be sure the inch button is lit.

Event 1 Press **TOOL 1001** and **EVENT ENTER.** This stores the displayed information into memory and advances to the next event.

NOTE:

If the wrong information is pressed before pressing EVENT ENTER, press EVENT CLEAR once. This will clear the event so the correct information can now be put in and entered.

Event 2: Event #2 on the program sheet shows an empty event (Tool Length Offset). Press **EVENT ENTER** and this will enter an empty event. The information for this event will be entered later when you set the tool offset for tool #1.

Event 3: Press **TOOL 1002** and press **EVENT ENTER.**

Event 4: Enter another empty event by pressing **EVENT ENTER** (this will be the offset value for tool #2).

Event 5: Press **TOOL O** and **EVENT ENTER.**

Event 6: Press **Z O.** The light for the **RAPID/FEED** button should be on for **RAPID.** (If the **FEED** button is lit, press the button once). The light should be off for **INCR.** (If the **INCR** button is lit, press the button once). This will show that the move is in **RAPID** (100 IPM) and is an absolute dimension. Press **EVENT ENTER** and enter this information.

Event 7: Press **X-3. Y1.** Notice the **RAPID** light is still on and the INCR light is still off. Press **EVENT ENTER**

Event 8: Press **TOOL 1 EVENT ENTER**

Event 9: Press **V20 6.** When the V key is pressed, the Y axis display will light. This is where the value of V20 is entered. After the two digits (20) following V are entered, the next digits (10.) are automatically entered in the Y axis display. Press **EVENT ENTER.**

Event 10: Press **V21 .1** press **EVENT ENTER.**

Event 11: Press **G** and **81**. The 81 will be shown in the upper right hand display. Press **EVENT ENTER.**

Event 12: Press **X .5 Y .5 Z -.130.** Leave the **RAPID** light on and the **INCR** light off. Press **EVENT ENTER.**

Event 13: Press **Y -3.** and leave the **RAPID** button lit. Press the **INCR** button to change the command to incremental. Press **EVENT ENTER.**

Event 14: Press **X 4.5** and leave **RAPID** on. Press **INCR** to make absolute active. Press **EVENT ENTER.**

Event 15: Press **Y 3.** and leave **RAPID** on. Press **INCR** to turn it on. Press **EVENT ENTER**.

Event 16: Press **G80 EVENT ENTER.**

Event 17: Press **TOOL O EVENT ENTER.**

Event 18: Press **Z O** leave **RAPID** on. Press **INCR** to turn it off. Press **EVENT ENTER.**

Event 19: Press **X -3. Y 1.** leave **RAPID** on and **INCR** off. Press **EVENT ENTER.**

Event 20: Press **TOOL 2 EVENT ENTER.**

Event 21: Press **V20 4.** Press **EVENT ENTER.**

Event 22: Press **V21 .1** press **EVENT ENTER.**

Event 23: Press **G 81 EVENT ENTER.**

Event 24: Press **X .500 Y -.500 Z -.35** leave the **RAPID** button on and **INCR** off. Press **EVENT ENTER.**

Event 25: Press **Y -3.** leave the **RAPID** button on, press **INCR** to turn it on. Press **EVENT ENTER.**

Event 26: **Press X 4.5** leave **RAPID** on and press **INCR** to turn it off. Press **EVENT ENTER.**

Event 27: Press **Y 3.** leave **RAPID** on and press **INCR** to turn it on. Press **EVENT ENTER**.

Event 28: Press **G80 EVENT ENTER.**

Event 29: Press **TOOL O EVENT ENTER.**

Event 30: Press **Z O** leave **RAPID** on and press **INCR** to turn it off. Press **EVENT ENTER.**

Event 31: Press **X -3. Y 1.** leave **RAPID** on and **INCR** off. Press **EVENT ENTER.**

Event 32: Press **END EVENT ENTER.**

After a program has been entered into the control, it should always be checked before attempting to run a part. To check the program, press **PROGRAM CHECK, 1, EVENT SEARCH.** If the information displayed is correct, press **START** and event #2 will be displayed. If it is correct, continue by pressing the **START** button to advance through and check the program. Compare the display of each event with the program sheet to ensure the information was entered correctly. By pressing the **HOLD** button in program check, the previous event will be displayed so that you can go back an event.

C. ADD EVENT

Sections 6-C and D will give detailed explanations of the Crusader editing capabilities. For an example of using **ADD EVENT**, let us say the G81 in event 11 of this exercise was not entered and by mistake we have entered the information for event 12 in event 11. It becomes necessary to add an event and shift the information in event 11 to event 12. To do this, the first step is to display event 11 in **PROGRAM ENTER** mode. Press **PROGRAM ENTER, 11, EVENT SEARCH.** At this time event 11 will be displayed.

Press **ADD EVENT** and this will shift event 11 (and all events following) up one event. (i.e. 12 becomes 13, 13 becomes 14, and so on for all the following events). This also creates an empty event in which the missing information can be entered. To do this, press **G81, EVENT ENTER.** The information has been entered into the empty event just created.

D. DELETE EVENT

If an empty event was entered or an unwanted event was entered by accident, it can be deleted. For example, let us say that the information in event 11 was entered twice (once in event 11 and once in event 12). To delete event 12, press **PROGRAM ENTER** mode, **12, EVENT SEARCH, DELETE EVENT.**

The information which was in event 12 is now removed and the information in event 13 will be shifted to event 12. (i.e. 14 becomes 13, 15 becomes 14 and so on for all the following events). Using these two features it is possible to add and delete information without rewriting the entire program.

E. MACHINE SET UP

The head of the machine should be trammed 90 degrees to the X and Y axis and the vise should be parallel to the X axis. A workpiece stop should be on the left side (see Figure 34).

Place the part in the vise and against the workpiece stop on parallels not thicker than 5/16". Put an edge finder in the spindle. Using the hand wheels or the manual JOG buttons, find the left edge of the part. Press **MANUAL,** then set zero by pressing **(X), O, EVENT ENTER.**

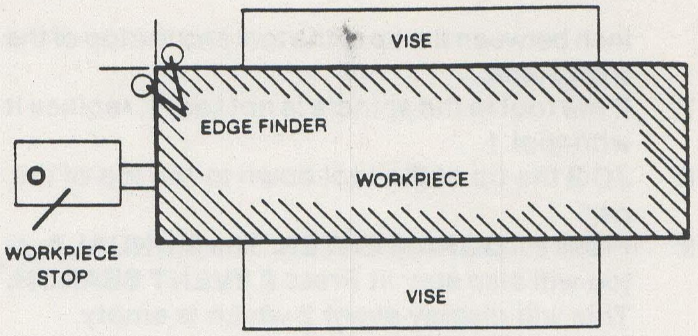


Figure 34: TOP VIEW OF WORKPIECE IN VISE

Raise the edge finder above the top of the workpiece and, using the readout display, move the edge finder a distance half of its diameter over the workpiece. Set zero again by pressing **(X), O, EVENT ENTER.** Move the edge finder to the top edge of the part. Set (Y) O by pressing **(Y) O EVENT ENTER.** Raise the edge finder above the top of the part and, using the readout display, move the edge finder a distance half of its diameter over the workpiece. Set (Y) O again by pressing **(Y), O, EVENT ENTER.** If you move over the top left corner of the part, the readout will display (X) O and (Y) O.

F. SETTING TOOL LENGTH OFFSETS

Tool offsets can be set with the motors off or on. If the motors are off, jog commands must be executed manually.

To set the tool length offsets for Programming Example 1, follow the step-by-step instructions below:

1. Put the spot drill and .250 drill each in their own quick change holder.
2. Turn on the servos and press **MANUAL,** even if MANUAL is lit, to clear any previous command.
3. **JOG** the quill up near the top limit. If the limit switch is tripped, lower the quill by hand until you are just off the top limit switch and re-start the servos.
4. Press **TOOL, O, START**, to make sure no offset is active.
5. Press **Z, O, EVENT ENTER** to set this as our Z, O full retract position. See figure 19 in section 4-J.
6. Place the longest tool in the spindle and set the table height using the knee. Leave about ¼

inch between the tip of the tool and the top of the workpiece.

7. If the tool in the spindle is not tool 1, replace it with tool 1.
8. **JOG** the tip of the tool down to the top of the part
9. Press **PROGRAM ENTER.** The **MANUAL** button will also stay lit Press **2,EVENT SEARCH.** This will display event 2 which is empty.
10. Press **DISP POSN.** This will turn on the display and show our absolute position. Press **Z, EVENT ENTER.** This will take the absolute position in the Z axis display and enter it as tool #1's offset in event 2. This is our distance of compensation for tool 1. The control now knows where the top of the part is when using tool I.
11. Using the **JOG** buttons, raise the tool off the workpiece and place tool 2 in the spindle.
12. **JOG** this tool down to the top of the part.
13. Press **4 EVENT SEARCH.** Press **Z EVENT ENTER.** This enters the absolute position of the Z axis as the offset for tool #2 in event 4.

G. REVIEW OF PROGRAMMING FORMAT

Let's review our procedure so far. This should be a standard format for each job to be done with any CNC. We first examined the print to find the best way to hold our part. We selected a point to set absolute zero, selected our tooling, and thought about a logical sequence. We wrote the program, using a format of retracting the quill and moving to a tool change position first.

Next, we entered the program into memory and checked it to see if it was entered correctly. X and Y zero was set, then the tool length offsets were entered. See figure 35.

If these steps have been followed correctly, we are ready to run the part in a dry run, SINGLE STEP mode.

H. RUNNING THE PROGRAM

Remove the part from the vise and be ready to press **EMERGENCY STOP** while doing this dry run.

Turn on the servo motors. Press the **SINGLE STEP** BUTTON. Then press **1 EVENT SEARCH.** This will insure that we start from the beginning of the program. Press the **START** button to execute events 1-6. Each time the **START** button is pressed the control will execute 1 move of the program (If AUX 1901 is active, refer to the AUX code listing in the miscellaneous section). When the sixth event is executed, the quill will retract. Event 7 will move the quill to the tool change position. Event 8 will display that the operator is to put tool 1 in the spindle (the program sheet tells us that the tool should run at 3,000 RPM).

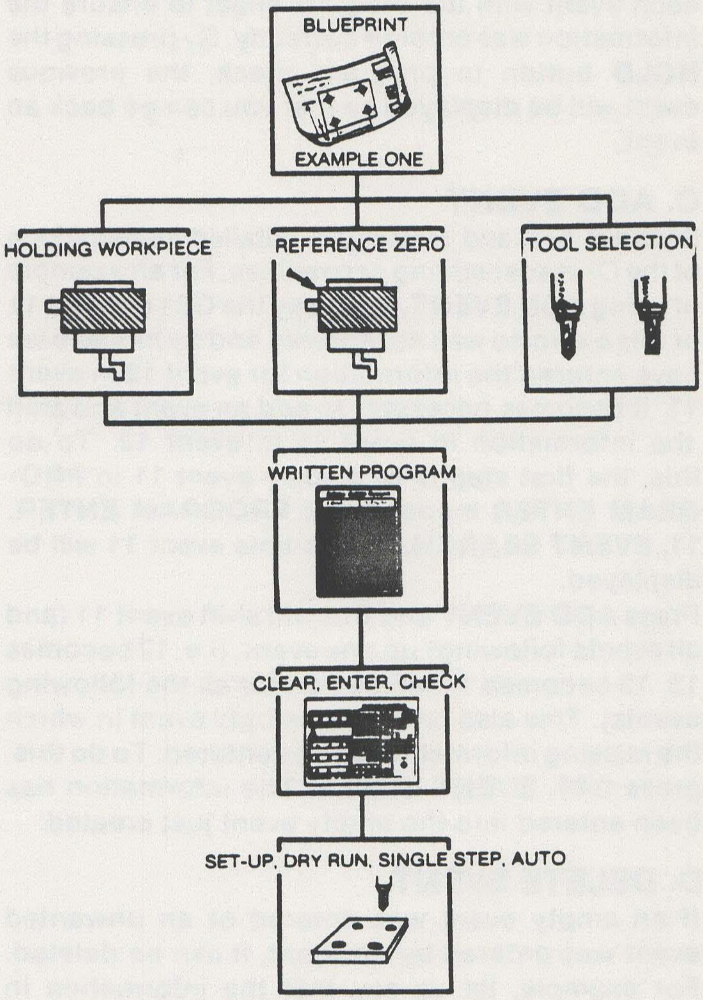


Figure 35: PROGRAMMING STEPS

Turn on the spindle and press **START** to proceed through each step of the program. Check each move to see that the machine moves to the correct position at the correct feedrate. When tool 2 is called for in the program, put it in the spindle and continue through the program. If errors are found, edit the program.

Once this is done, put the machine in **AUTO** and put the part in the vise. Press **1 EVENT SEARCH START** and the quill will retract, move to the tool change position and stop. Put tool 1 in the spindle, turn it on and press **start** to make the part

SECTION 7:
DRILLING EXAMPLES

This chapter will explain in detail how the operator may use the Crusader control to drill into a workpiece. Several features and functions are explained; such as peck drilling, do-loops, subroutines, etc.

A. DRILLING FORMAT RULES

When programming the G80 drilling cycles, these are six basic rules that must be followed. The example in figure 36 will be used to illustrate these rules in practical form.

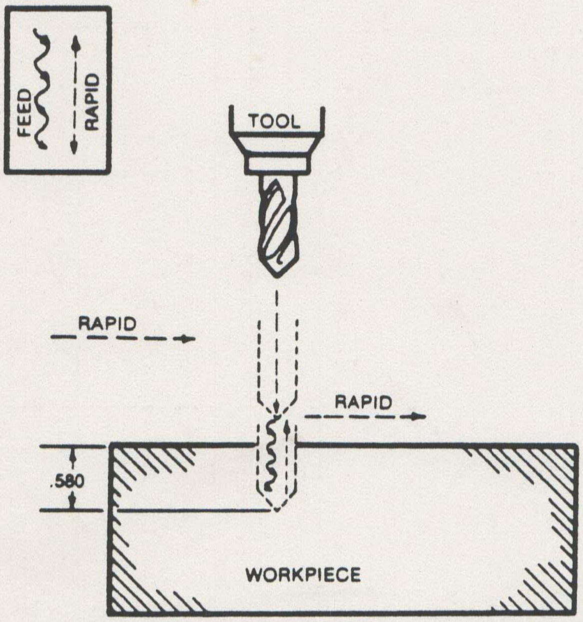
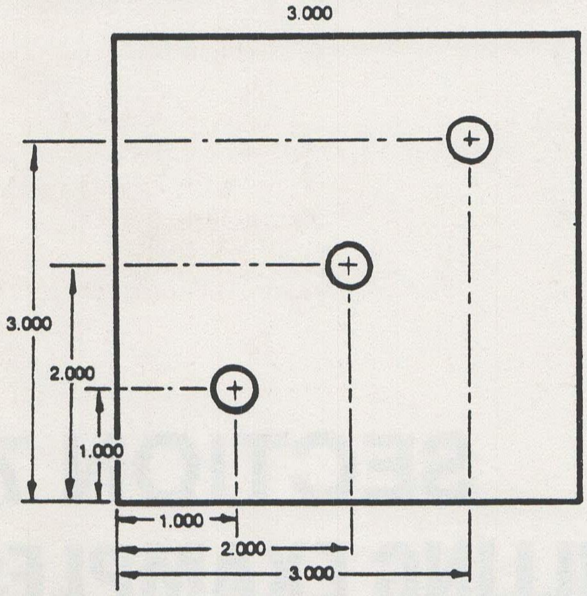


Figure 36: BASIC DRILLING FORMAT RULES



EVENT:	FUNCTION:
1. Tool 1001	Offset for Tool 1
2.	Offset value entered at set up
3. Tool 0	De-activate offsets
4. Z 0 RA	Retract quill
5. X-1. Y1. RA	Rapid to a tool change position
6. Tool 1	Activate Tool 1 offset, program stops
7. V20 8.	Set drilling feedrate
8. V21 .100	Set drilling start height
9. G81	Activate drilling
10. X1. Y1. Z-.58 RA	First hole location and Z depth
11. X2. Y2.	Second hole location
12. X3. Y3.	Third hole location
13. G80	De-activate drilling
14. Tool 0	De-activate tool offset
15. ZO RA	Retract quill
16. X-1. Y1. RA	Rapid to tool change position
17. End	End program

DRILLING FORMAT RULES

1. Program a feedrate as a **V20.** For example, **V20** 8. will be a feedrate of 8 inches per minute when using the G80 series.
2. Program a dimension in **V21** as the height which you want the tool to start feeding into the workpiece. This will also be the point which the tool will retract to. If G83 or G87 is used, this dimension must be .100 above the surface which will be drilled into. **V21** .1 will set the distance at .100 above zero. This dimension must always be an absolute value.
3. If G82 or G89 is going to be used, program **V22** and the time (in seconds) that the tool will dwell at the bottom of the hole. **V22** 1.5 will cause a 1.5 second dwell at the bottom of each hole when using G82 or G89. If any other G80 series code is used, a dwell is not performed, therefore, **V22** does not need to be entered. If entered, **V22** must be positive.
4. If a pecking cycle G83 or G87 is used, the maximum peck is given as **V23.** The control will make each peck equal and will not exceed the depth given. **V23** .160 will set the maximum peck at .160. This distance is given as a positive incremental value. The incremental light is not used, but the control interprets the V23 dimension as incremental.
5. Program the appropriate G code. A listing of all G80 series codes and a description of each is given in Section 4-M.
6. Program the X Y coordinates and Z depth of the first hole. This move should be in rapid. When this event is executed, the control will rapid in X and Y to the programmed position, rapid the Z axis to the point given in **V21** as the height to start drilling from. It will then drill the hole and retract to the starting height.

List the X and/or Y positions for the remaining holes to be drilled. Each time a new position is reached, the G code will become active and the hole will be drilled. After the last hole, program a G80 to deactivate the drilling cycle. Using event search or changing modes (except changing from single step to auto) will also deactivate any drilling cycle.

When canned cycles are being executed, disregard the event number being displayed. The event numbers displayed are events that the internal software uses.

If a feed rate is not entered or your maximum peck is larger than the total depth drilled, the machine will dwell upon execution of the canned cycle. If the machine does this, check your program for an error in **Program Check.**

When peck drilling is used, the control will assume that the V 21 dimension is .100 (2. mm in metric) above the top of the workpiece. The control then adds this distance to the first peck so that all pecks are of equal value and all pecks are into the workpiece. To change this distance, program an event using V99 and the value desired. For example when machining rough castings a larger value may be needed. If .375 is the desired distance, program V99 .375 as an event before activating the G code for drilling. This value will stay active unless changed back by programming V99 .100, or if the control is turned off, or if switched from inch to mm.

B EXAMPLE 2 (DO LOOPS)

Figure 37 shows a part with equally spaced holes. The same procedures outlined in section A should be followed for each part. The first 3 steps again will be 1) how the part is held, 2) where zero will be set, 3) and what tooling will be used. Since the part is small and rectangular, hold it in a vise. Absolute zero will be set at the upper left corner Tool 1 will be a spot drill, Tool #2 will be a .250 diameter drill.

The program will first list the offsets, then retract the quill and move to the tool change position. The first hole will be spot drilled using G81. The remaining 4 holes in the first row will be drilled using a do-loop. Each successive row of holes will be drilled using a do-loop. After all holes are center-drilled, the G code will be deactivated, the quill retracted and moved to the tool change position. Tool 2 will drill the holes, again using G81 and a series of do-loops. Refer to figure 37 as you go through the program.

EXAMPLE 2

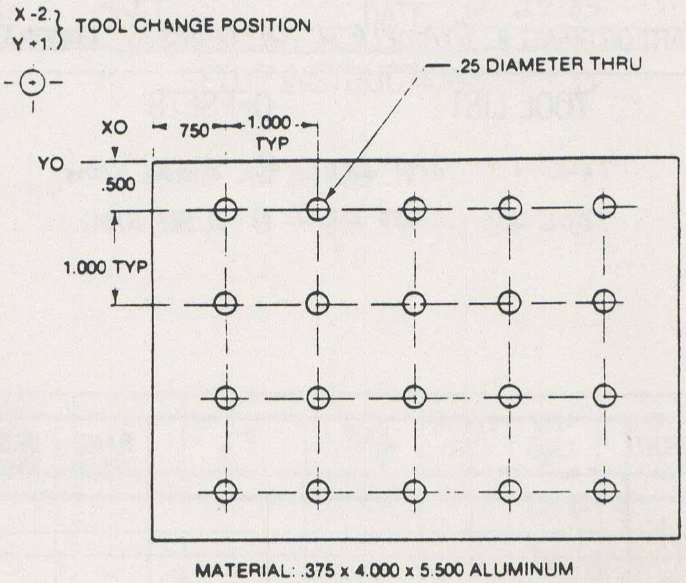


Figure 37: DO LOOPS

EVENT:	**FUNCTION**
1. TOOL 1001	Set offset #1
2.	Open event for value of offset #1
3. TOOL 1002	Set offset #2
4.	Open event for value of offset #2
5. TOOL O	Deactivate offsets
6. ZO RA	Retract quill.
7. X-2. Y1. RA	Tool change position
8. TOOL 1	Activate offset #1.
9. V20 14.	Set the feedrate at 14. IPM.
10. V21.1	Sets the drilling start height.
11. G81	Activate drilling cycle for spot drill
12. X.75 Y-.500 Z-.14 R A	The X Y location and Z depth of the first hole. The event is programmed in rapid but because G81 is active, the Z axis will rapid to the start height then feed to the Z axis depth.
13. DO 4	Do loop for the 1st line of 4 holes.
14. X1. RI	Incremental distance between holes (G 81 is active to drill each increment). The do loop will cause 4 holes to be spot drilled at 1. increments.
15. END	End for do loop. The program will continue because this event only ends the do loop.

SHEET 1 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 2: DO-LOOPS PROGRAMMER RJT DATE 12·27·82

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
TOOL 1: SPOT DRILL @ 3,500 RPMs TOOL 2: .250 DRILL @ 3,500 RPMs		USE VISE WITH STOP ON LEFT SIDE OF WORKPIECE. X, Y "0" SET AT UPPER LEFT CORNER.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2												
3									1002			
4												
5									0			
6			0	R	A							
7	-2.	1.		R	A							
8									1			
9								V20: 14.				
10								V21: .1				
11							81					
12	.750	-.500	-.140	R	A							
13											DO4	
14	1.			R	I							
15												END
16		-1.		R	I							
17											DO4	
18	-1.			R	I							
19												END
20		-1.		R	I							

SHEET 2 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 2: DO-LOOPS PROGRAMMER RJT DATE 12·27·83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.		SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
21											D04	
22	1.			R	I							
23												END
24		-1.		R	I							
25											D04	
26	-1.			R	I							
27												END
28							80					
29									0			
30			0	R	A							
31	-2.	1.		R	A							
32									2			
33								V20: 10.				
34								V21: .1				
35							81					
36	.750	-.500	-.500	R	A							
37											D04	
38	1.			R	I							
39												END
40		-1.		R	I							

SHEET 3 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 2: DO-LOOPS PROGRAMMER RJT DATE 12·27·83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.		SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
41											DO 4	
42	-1.			R	I							
43												END
44		-1.		R	I							
45											DO4	
46	1.			R	I							
47												END
48		-1.		R	I							
49											DO4	
50	-1.			R	I							
51												END
52							80					
53									0			
54.			0	R	A							
55	-2.	1.		R	A							
56												END
7												
8												
9												
0												

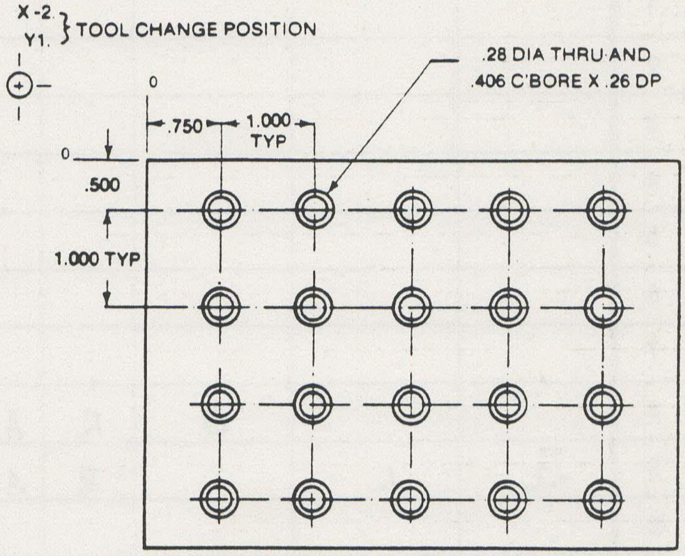
16. Y-1.0 R I	Incremental move to next line (G 81 still active so first hole is spot drilled).
17. DO 4	Do loop for next line of holes.
18. X-1.0 R I	Incremental distance in the negative direction.
19. END	End for do loop.
20. Y-1. R I	Incremental move to the third line of holes. (G81 still active so first hole is spot drilled.)
21. DO 4	Do loop for line of holes.
22. X1.0 R I	Incremental distance
23. END	End of do loop
24. Y-1.0 R I	Incremental move to last line of holes
25. DO 4	Do loop for last line.
26. X-1.0 R I	Incremental distance in the negative direction.
27. END	End of do loop .
28. G 80	Deactivate drilling cycle.
29. TOOL O	Deactivate tool offset
30. ZO R A	Retract quill
31. X-2. Y1. R A	Tool change position
32. TOOL 2	Activate offset for tool #2, stops program
33. V20 10.	Sets feedrate at 10 IPM.
34. V21 .1	Sets drilling start height
35. G 81	Activate drilling cycle.
36. X.750 Y-.500 Z-.500 R A	Is the X Y location and Z depth of the first hole.
37. DO 4	Do loop
38. X1.0 R I	Increment. (G 81 active drills 4 holes at 1. increments.)
39. END	End for do loop.
40. Y-1. R I	Incremental move and drills first hole in next run.
41. DO 4	Do loop
42. X-1.0. R I	Increment
43. END	End do loop
44. Y-1.0 R I	Incremental move down.
45. DO 4	Do loop
46. X1.0 R I	Increment
47. END	End do loop
48. Y-1.0 R I	Incremental move down
49. DO 4	Do loop
50. X-1.0 R I	Increment
51. END	End do loop
52. G 80	Deactivate drilling cycle
53. TOOL O	Deactivate tool offset
54. ZO RA	Retract quill
55. X-2. Y1. RA	Tool change position
56. END	End of program.

Enter this program and see how the do loops work with the drilling cycle. It will be good practice.

C. EXAMPLE #3

Figure 38 shows how to apply a subroutine to the same program. In the last program only 2 tools were used. With this program we will use the 3 tools that are described on the tool sheet. However, by grouping do loops in a subroutine the program will be shorter than the last example even though you added another tool.

Subroutine 1 will be the series of do loops used to position the tools to the hole locations. The same subroutine will be used to spot drill, drill and counterbore the holes. Use figure 38 for reference as you examine the program.



MATERIAL: .375 x 4.000 x 5.500 ALUMINUM

Figure 38: SUBROUTINES

When writing a subroutine ensure the information in the sub has only the things you want repeated. As an example, this sub will be used with all the tools. Each tool goes to the same location but each will have a different feedrate, will use a different canned cycle and will machine to a different depth. So these are things that you would not put into the subroutine.

The things that will be repeated are: the do loops for each line of holes; the incremental move to each line of holes; and cancellation of the canned cycle when all the positioning moves are complete. No matter what tool you are using, you will need to deactivate the tool offset, retract the quill and move to the tool change position. Because these instructions will be repeated with each tool, these instructions will be in the sub. Since you already wrote the program without sub's, turn back to Example 2 and look at events

SHEET 1 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 3: SUBROUTINES PROGRAMMER MWA DATE 9·16·83

TOOL LIST OFFSETS

TOOL 1: SPOT DRILL @ 3,500 RPMS
TOOL 2: .28 DRILL @ 3,000 RPMS
TOOL 3: .406 ENDMILL @ 2,500 RPMS

SETUP INSTRUCTIONS

X, Y "0" SET IN UPPER LEFT CORNER.
PART IS HELD IN VISE WITH STOP ON LEFT SIDE.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2												
3									1002			
4												
5									1003			
6												
7									0			
8			0	R	A							
9	-2.	1.		R	A							
10									1			
11								V20: 15.				
12								V21: .1				
13							81					
14.			-.14	R	A							
15											CALL 1	
16									2			
17								V20: 10.				
18								V21: .1				
19							81					
20			-.5	R	A							

ANILAM **Anilam Electronics Corporation** 5625 N.W. 79th Avenue, Miami, Florida 33166 / (305) 592-2727 / Telex 26-4024

SHEET 2 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 3: SUBROUTINES PROGRAMMER MWA DATE 9·16·83

TOOL LIST OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.	SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
21											CALL 1	
22									3			
23								V20: 8.				
24								V21: .1				
25								V22: 2.				
26							82					
27			-.26	R	A							
28											CALL 1	
29												END
30												SUB 1
31	.750	-.500		R	A							
32											DO 4	
33	1.			R	I							
34.												END
35		-1.		R	I							
36											DO 4	
37	-1.			R	I							
38												END
39		-1.		R	I							
40											DO 4	

SHEET 3 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 3: SUBROUTINES PROGRAMMER MWA DATE 9·16·83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.		SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
41	1.			R	I							
42												END
43		-1.		R	I							
44											DO 4	
45	-1.			R	I							
46												END
47							80					
48									0			
49			0	R	A							
50	-2.	1.		R	A							
51												END
2												
3												
4.												
5												
6												
7												
8												
9												
0												

13 through 31 and 37 through 55. They are exactly the same. That is what the subroutine will look like. One more thing to remember. A subroutine is always entered into the events that follow the main program. If the main program is from event 1 through event 29, the subroutine would be entered at event 30. To search for a subroutine when editing, press **EVENT CLEAR** to clear any previous command, then press **SUBR 1, EVENT SEARCH.**

1. TOOL 1001	Set offset #1
2.	Open event for value of offset #1
3. TOOL 1002	Set offset #2
4.	Open event for value of offset #2
5. TOOL 1003	Set offset #3
6.	Open event for value of offset #3.
7. TOOL 0	Deactivate any tool offset.
8. ZO RA	Retract quill.
9. X-2. Y1. RA	Move to tool change position.
10. TOOL 1	Activate offset #1.
11. V20 15.	Sets spot drilling rate at 15. IPM.
12. V21.1	Set drilling start height.
13. G 81	Activates canned drilling cycle.
14. Z-.140 RA	Depth to be spot drilled. The G81 drilling cycle will not take effect until an X and/or Y coordinate is executed.
15. CALL 1	Call subroutine #1. This will spot drill all holes because G 81 is active. Go to sub 1 and follow the program to see how the holes will be spot-drilled. After the sub is executed, the program will return to Event 16.
16. TOOL 2	Activate offset #2.
17. V20 10.	Sets drilling feedrate at 10. IPM.
18. V21.1	Sets drilling start height
19. G81	Activates canned drilling cycle.
20. Z-.5 RA	Depth to be drilled.
21. CALL 1	Call subroutine #1. This drills all holes because G81 is active.
22. TOOL 3	Activate offset #3
23. V20 8.	Sets counterboring feedrate at 8 IPM.
24. V21.1	Sets counterboring start depth.
25. V22 2.	Sets dwell time at 2 seconds at bottom of counterbore.
26. G 82	Canned counter boring cycle.
27. Z-.260 RA	Depth to be counter bored.
28. Call 1	Call subroutine #1. This will counter-bore all holes because G82 is active.
29. END	End of program.
30. SUB 1	Label this pattern as subroutine 1.
31. X.750 Y-.500 RA	First X Y location in pattern.
32. DO 4	Do loop for 4 holes.
33. X1.0 RI	Incremental distance across X axis between holes.
34. END	End for do loop.
35. Y-1.0 RI	Incremental distance in Y axis to next line of holes.
36. DO 4	Do loop for 4 holes.
37. X-1.0 RI	Negative incremental distance across X axis between holes.
38. END	End for do loop.
39. Y-1.0 RI	Incremental distance in Y axis to next line of holes.
40. DO 4	Do loop for 4 holes.
41. X1.0 RI	Incremental distance across X axis between holes.
42. END	End for do loop
43. Y-1.0 RI	Incremental distance in Y axis to next line of holes.
44. DO 4	Do loop for 4 holes.
45. X-1.0 RI	Negative incremental distance across X axis between holes.
46. END	End for do loop.
47. G 80	Deactivate canned drilling cycle.
48. TOOL O	Deactivate tool offset.
49. ZO RA	Retract quill.
50. X-2. Y1. RA	Move to tool change position.
51. END	End of subroutine

By using subroutines, you cut the program from 56 events (using 2 tools) to 51 events (using 3 tools). You saved 5 events and did one completely new operation.

D. NESTING

In this program, we used "nesting". Nesting is putting a pattern inside of another pattern. Our do loops

SHEET 1 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 4: PECK DRILLING PROGRAMMER DFF DATE 11·9·83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
TOOL 1: SPOT DRILL @ 3,500 RPM's TOOL 2: .250 DRILL @ 3,000 RPM's TOOL 3: .375 DRILL @ 2,500 RPM's		PART HELD IN VISE WITH STOP ON LEFT SIDE. X, Y "0" AT TOP LEFT CORNER. PARALLELS 5/32" THICK MAX.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2												
3									1002			
4												
5									1003			
6												
7									0			
8			0	R	A							
9	-2.	1.		R	A							
10									1			
11								V20: 15.				
12								V21: .1				
13							81					
14.			-.2	R	A							
15											CALL 2	
16			-.135									
17											CALL 1	
18									2			
19								V20: 8.				
20								V21: 1.				

ANILAM **Anilam Electronics Corporation** 5625 N.W. 79th Avenue, Miami, Florida 33166 / (305) 592-2727 / Telex 26-4024

SHEET 2 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 4: PECK DRILLING PROGRAMMER DFF DATE 11·9·83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.		SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
21								V23: .15				
22							83					
23			-1.15	R	A							
24											CALL 1	
25									3			
26								V20: 10.				
27								V21: .1				
28								V23: .2				
29							83					
30			-1.2	R	A							
31											CALL 2	
32							80					
33									0			
34.			0	R	A							
35	-2.	1.		R	A							
36												END
37												SUB 1
38	2.750	-.375		R	A							
39											DO 5	
40		-.500		R	I							

SHEET 3 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 4: PECK DRILLING PROGRAMMER DFF DATE 11·9·83

TOOL LIST OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.	SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
41												END
42							80					
43									0			
44			0	R	A							
45	-2.	1.		R	A							
46												END
47												SUB 2
48	1.	-.375		R	A							
49	.500	-2.		R	A							
50	1.	-3.125		R	A							
51	4.			R	A							
52	4.5	-2.		R	A							
53	4.	-.375		R	A							
54.												END
5												
6												
7												
8												
9												
0												

were the first patterns. But by grouping them in a subroutine, we put our first pattern into a sub pattern that can be called anytime we need it.

Putting do loops into subroutines is one form of nesting. Another form is to call a subroutine and in that subroutine call a 2nd subroutine. There is no limit how far you can nest with the Crusader.

E. EXAMPLE 4: PECK DRILLING

In this example, you will use the G83 (Peck drilling) code to make the part shown in figure 39. The middle row of holes will be listed in subroutine 1. The outside pattern will be listed as subroutine 2.

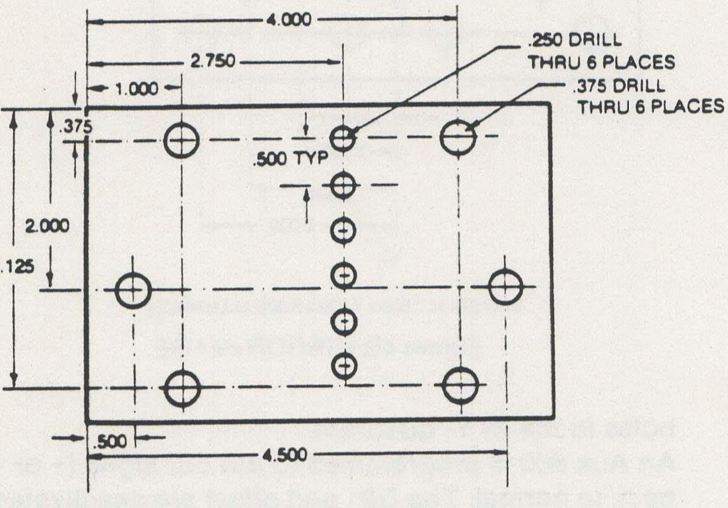


Figure 39: PECK DRILLING

A spot-drill (tool 1) will use both subroutines and will spot-drill each hole using the G 81 code. The second tool, a .250 drill (tool 2), will use subroutine 1 and will peck drill the center row of holes using G83. A .375 drill (tool 3) will use subroutine 2 and will peck drill the outside pattern of holes using the G 83 code.

An explanation of each event for this program is listed below:

1 TOOL 1001	Set offset #1.
2.	Open event for value of offset #1.
3. TOOL 1002	Set offset #2.
4.	Open event for value of offset #2.
5. TOOL 1003	Set offset #3.
6.	Open event for value of offset #3.
7. TOOL O	Deactivates offset.
8. ZO RA	Retracts quill.
9. X-2. Y1. RA	Rapids to tool change position.
10. TOOL 1	Activate tool 1's offset, stops program.
11. V20. 15.	Sets drilling feedrate.
12. V21 .1	Sets starting height.
13. G 81	Activates canned cycle.
14. Z-.2 RA	Drilling depth.
15. CALL 2	Executes sub 2, spot drills outside hole pattern.
16. Z-.135	Sets new drill depth for middle row of holes.
17. CALL 1	Execute sub 1, spot drill middle row of holes, cancels drilling, cancels tool offset, moves to tool change.
18. TOOL 2	Activates tool 2 offset, stops program.
19. V20 8.	Sets feedrate at 8 IPM.
20. V21 .1	Sets starting height.
21. V23 .15	Sets maximum peck at .15 deep. The control will calculate how deep to make each peck so that each one is an equal depth and does not exceed the total depth.
22. G 83	Activates canned peck drilling cycle. The tool will retract to the starting height after each peck.
23. Z-1.15 RA	Sets drilling depth.
24. CALL 1	Executes sub 1 which peck drills middle row of holes, deactivates G83, deactivates tool length offset, retracts quill and moves to tool change position.
25. TOOL 3	Activates tool 3 offset, stops program.
26. V20 10.	Sets feedrate at 10 IPM.
27. V21 .1	Sets starting height at .1
28. V23 .2	Sets max peck at .2.
29. G 83	Activates canned peck drilling.
30. Z-1.2 RA	Sets drilling depth.
31. CALL 2	Executes sub 2 which peck drills outside pattern of holes.
32. G80	Deactivates G 83.
33. TOOL O	Deactivates tool length offset.
34. ZO RA	Retracts quill.
35. X-2. Y1. RA	Rapids to tool change location.
36. END	End of program

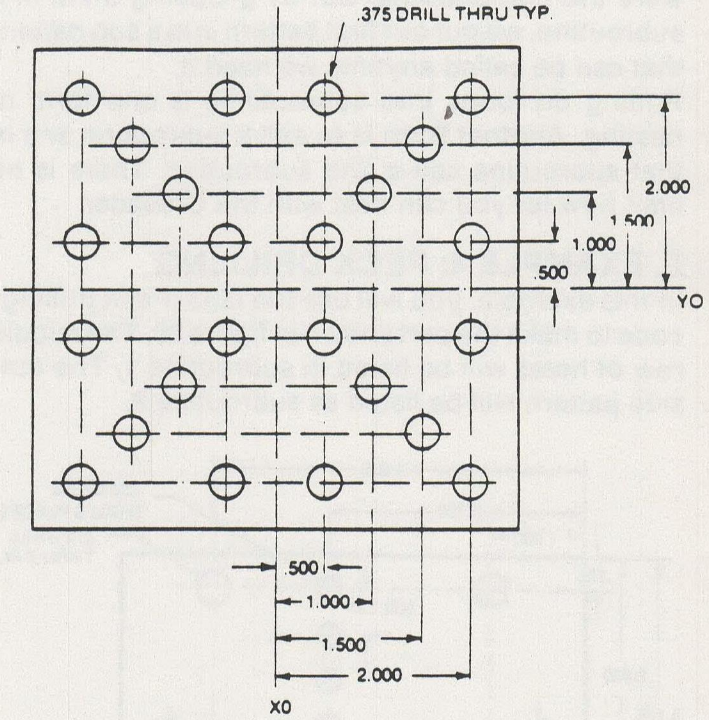
37. SUB 1	Start of subroutine 1.
38. X2.750 Y-.375 RA	Moves to top hole of middle row.
39. DO 5	Sets do loop to be repeated 5 times.
40. Y-.500 RI	Incremental distance between holes.
41. END	End of do loop.
42. G 80	Deactivates canned drilling cycle.
43. TOOL O	Deactivates tool length off set.
44. ZO RA	Retracts quill.
45. X-2. Y1. RA	Rapids to tool change position.
46. END	End of subroutine 1.
47. SUB 2	Start of subroutine 2.
48. X1. Y-.375 RA	Events 48-53 list
49. X.500 Y-2. RA	X Y locations of the outside
50. X1. Y-3.125 RA	pattern of holes.
51. X4. RA	
52. X4.5 Y-2. RA	
53. X4. Y-.375 RA	
54. END	End of subroutine 2.

F. EXAMPLE 5: MIRROR IMAGE

Mirror image is used to change the sign of a dimension, such as changing X+ to X-. This feature is helpful for many applications such as left and right handed parts or mold making. The part shown for Example #5 (Figure 40) has a hole pattern that is the same on each side of the zero reference point. By writing a subroutine with the hole locations for one quarter, that pattern can be flipped using AUX codes and changing our X dimensions or Y dimensions (or X and Y together) from plus to minus or minus to plus. A complete AUX code listing is found in the Miscellaneous section.

The program for Example 5 will be written using 2 subs, the first will have all the hole locations in the upper right quadrant. This quadrant is chosen because all the dimensions are X+ Y+ and you do not have to remember to put a minus sign in. (See figure 6 if quadrants are not fully understood.)

Sub 2 will call Sub 1 which will position the tool to all the locations in the upper right quadrant. Sub 2 will then activate Aux 100 to mirror all the X+ dimensions to X- and call Sub 1 again. This will drill the holes in the X- Y+ quadrant. Aux 200 is then to be programmed to mirror the pattern in the Y axis. Again, Sub 1 will be called. This will drill the holes in the X+ Y- quadrant. Next we must mirror the original pattern across X and Y and again call Sub 1. This drills the holes in the X- Y- quadrant.



MATERIAL: .500 x 5.000 x 5.000 ALUMINUM

Figure 40: MIRROR IMAGE

An Aux 800 is programmed to put our signs (+ or -) back to normal. The G81 and offset are deactivated, the quill is retracted and sent to tool change position. In this way we can use Sub 2 to spot drill then drill the entire part.

1.	TOOL 1001	Sets offset #1
2.		Open event for value of offset 1.
3.	TOOL 1002	Sets offset #2.
4.		Open event for value of offset.
5.	TOOL O	Deactivates tool offsets.
6.	ZO RA	Retracts quill.
7.	X-3.5 Y0. RA	Rapid to tool change position.
8.	TOOL 1	Activates tool 1 offset, program stops.
9.	V20 15.	Sets drilling feedrate.
10.	V21 .1	Set drilling start height.
11.	G81	Activates drilling canned cycle.
12.	Z-.135	Sets drilling depth.
13.	CALL 2	Executes Sub 2 which mirrors sub 1 to drill entire part. See sub 2 for details.
14.	TOOL 2	Activates tool 2 offset.

SHEET 1 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 5: MIRROR IMAGE PROGRAMMER BJT DATE 2·13·83

TOOL LIST OFFSETS	SETUP INSTRUCTIONS
TOOL 1: SPOT DRILL @ 3,000 RPM's TOOL 2: .375 DRILL @ 2,500 RPM's	PART IS HELD IN VISE WITH STOP ON LEFT SIDE. X, Y "0" IS AT THE CENTER OF THE PART.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2												
3									1002			
4												
5									0			
6			0	R	A							
7	-3.5	0		R	A							
8									1			
9								V20: 15.				
10								V21: .1				
11							81					
12			-.135	R	A							
13											CALL 2	
14.									2			
15								V20: 15.				
16								V21: .1				
17								V23: .2				
18							83					
19			-.65	R	A							
20											CALL 2	

SHEET 2 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 5: MIRROR IMAGE PROGRAMMER BJT DATE 2·13·83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.		SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
21												END
22												SUB 1
23	.500	2.000		R	A							
24	2.000			R	A							
25	1.500	1.500		R	A							
26	1.	1.		R	A							
27	.500	.500		R	A							
28	2.			R	A							
29												END
30												SUB 2
31											CALL 1	
32										AUX 100		
33											CALL 1	
34.										AUX 200		
35											CALL 1	
36										AUX 300		
37											CALL 1	
38										AUX 800		
39							80					
40									0			

SHEET 3 OF 3

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 5: MIRROR IMAGE PROGRAMMER BJT DATE 2·13·83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.		SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
41			0	R	A							
42	-3.500	0		R	A							
43												END
4												
5												
6												
7												
8												
9												
0												
1												
2												
3												
4.												
5												
6												
7												
8												
9												
0												

15. V20 15.	Sets feedrate at 15 IPM.
16. V21 .1	Sets drilling - start height
17. V23 .2	Sets maximum peck distance.
18. G83	Activates peck drill cycle.
19. Z-.65 RA	Sets drilling depth.
20. CALL 2	Executes sub 2 to peck drill part.
21. END	End of program
22. Sub 1	Start of sub 1.
23. X.500 Y 2. RA	Events 23-28 list the X and/or Y locations for the holes. in the X+ Y+ quadrant.
24. X2. RA	
25. X1.500 Y1.500 RA	
26. X1. Y1. RA	
27. X.500 Y.500 RA	
28. X2. RA	
29. END	End of sub 1.
30. Sub 2	Start of sub 2.
31. CALL 1	Executes sub 1 which drills holes in X+ Y+ quadrant.
32. AUX 100	Instructs the control to change the sign of X axis dimensions from here on.
33. CALL 1	Executes sub 1 with Aux 100 active, drills holes in X- Y+ quadrant.
34. AUX 200	Changes the sign of Y axis commands.
35. CALL 1	Executes sub 1 with Aux 200 active. Drills holes in X+ Y- quadrant.
36. AUX 300	Changes the sign of X and Y commands.
37. CALL 1	Executes sub 1 with Aux 300 active. Drills holes in X- Y- quadrant.
38. AUX 800	Deactivates mirror image.
39. G80	Deactivates drilling cycle.
40. TOOL O	Deactivates tool length offset.
41. ZO RA	Retracts quill.
42. X-3.500 YO RA	Rapids to tool change position.
43. END	End of sub 2.

SECTION 8 : MILLING EXAMPLES

It is necessary to understand the 6 circular milling format rules that are used with the Crusader. These rules have already been explained in Section 5-I. Review that information prior to getting involved in the following examples.

A. EXAMPLE 6: CIRCULAR MILLING

Tool #1 is a .500 end mill. It will be used at 1800 RPM with a feedrate of 10 IPM. The part is screwed to a fixture plate, X and Y zero is set in the lower left hand corner of the part.

> **NOTE**
> Remember that you are programming the center of the tool so the **programmed radius of each corner will be the part radius + or − the cutter radius.**

You will use the upper left hand corner for the start point and will climb mill around the part. The tool will be positioned so the cutting edge is **on** the top edge of the part and .100 off the left side of the part. (This is for clearance so the tool does not crash into the side of the workpiece.) This is Milling Position 1.

Each milling position is numbered in the order in which you will machine the part. Review figure 41 and use it for reference as you write the program.

The set-up and tool list are put at the top of the programming sheet. Events 1-6 are the standard format for starting a program. The offsets are listed, the quill is retracted and moved to the tool change position. Tool 1's offset is activated.

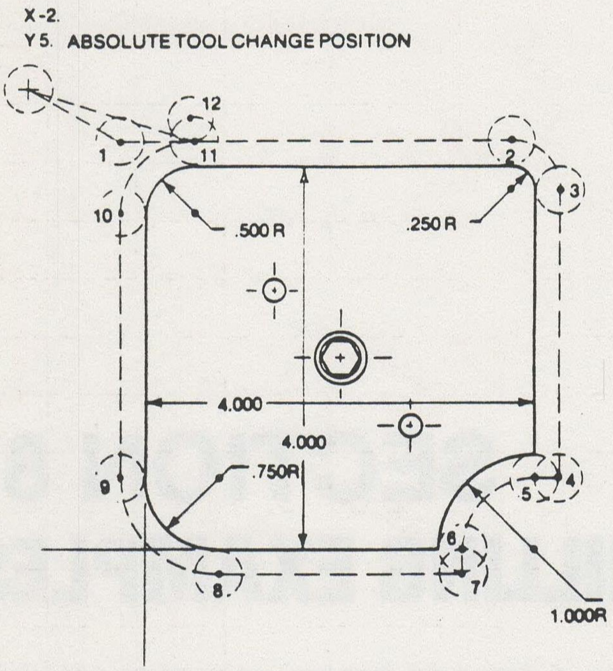


Figure 41: CIRCULAR MOTION

Event 7:	Moves the tool in rapid to position 1.
Event 8:	Rapids the tool to .100 above the top of the part.
Event 9:	Sets the feedrate to 10.0 IPM.
Event 10:	Feeds the tool to cutting depth.
Event 11:	Feeds the tool to position 2 (rules 1 and 2 for circular milling from Section 5-1).
Event 12:	Describes arc and direction CW (rule 3).
Event 13:	Describes the arc center (rule 4).
Event 14:	Describes the finish point of the arc (rule 5).
Event 15:	The last arc event (rule 6).
Event 16:	Feeds the tool to position 4.
Event 17:	Feeds the tool to position 5. This is needed to move the tool to the start point of the arc (rule 1 and 2).
Event 18:	Describes arc and direction CCW (rule 3).
Event 19:	Describes the arc center. It is described incrementally from the start point. The center point and end point **must** be defined as a 2 axis position (rule 4).
Event 20:	Describes the finish point of the arc (rule 5).
Event 21:	The last arc event.
Event 22:	Feeds the tool to position 7.
Event 23:	Feeds the tool to position 8.
Event 24:	Describes arc and direction CW.
Event 25:	Describes the arc center.
Event 26:	Describes the arc finish point.
Event 27:	The last arc event.
Event 28:	Feeds the tool to position 10.
Event 29:	Describes arc and direction CW
Event 30:	Describes the arc center.
Event 31:	Describes the arc finish point.
Event 32:	The last arc event.
Event 33:	An incremental move to get the tool off the workpiece.
Event 34:	Deactivates tool 1's offset.
Event 35:	Retracts the quill.
Event 36:	Rapids the tool back to the tool change position.
Event 37:	Ends the program.

Enter and check this program into memory. You can put a sheet of paper on the table and a pen in the spindle to draw the tool path to check it. To do this, set the tool offset when the pen is on the paper and change event 10 to Z, O; F, A so you do not drive the pen .550 into the paper.

SHEET 1 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 6: CIRCULAR MILLING PROGRAMMER JTW DATE 10.31.83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
TOOL 1: .500 END MILL @ 1,800 RPMs		SCREW PART TO FIXTURE. SET X,Y "0" IN LOWER LEFT CORNER OF PART.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2												
3									0			
4			0	R	A							
5	-2.	5.		R	A							
6									1			
7	-.350	4.250		R	A							
8			.100	R	A							
9				10.								
10			-.500	F	A							
11	3.75			F	A							
12						CW						
13	3.75	3.75		F	A							
14	4.25	3.75		F	A							
15						ARC						
16		.750		F	A							
17	4.			F	A							
18						CCW						
19	0	-.750		F	I							
20	-.750	-.750		F	I							

SHEET 2 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 6: CIRCULAR MILLING PROGRAMMER JTW DATE 10.31.83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.		SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
21						ARC						
22		-.250		F	A							
23	.750			F	A							
24						CW						
25	.750	.750		F	A							
26	-.250	.750		F	A							
27						ARC						
28		3.500		F	A							
29						CW						
30	.500	3.500		F	A							
31	.500	4.250		F	A							
32						ARC						
33		.100		F	I							
34.									0			
35			0	R	A							
36	-2.	5.		R	A							
37												END
8												
9												
0												

SHEET 1 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 7: MIRROR IMAGE PROGRAMMER BFD DATE 11-16-83

TOOL LIST OFFSETS	SETUP INSTRUCTIONS
TOOL 1: .500 BOTTOM-CUTTING END MILL @ 1,600 RPM'S	PLACE PART IN VISE WITH LOCATOR ON THE LEFT SIDE. SET X, Y "0" IN UPPER LEFT CORNER OF PART.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2												
3									0			
4			0	R	A							
5	-2.	1.		R	A							
6									1			
7	1.625	-.375		R	A							
8			.100	R	A							
9											CALL 1	
10	2.375	-.375		R	A							
11										AUX 100		
12											CALL 1	
13	2.375	-3.625		R	A							
14.										AUX 300		
15											CALL 1	
16	1.625	-3.625		R	A							
17										AUX 200		
18											CALL 1	
19									0			
20			0	R	A							

SHEET 2 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 7: MIRROR IMAGE PROGRAMMER BFD DATE 11·16·83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.		SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
21	-2.	1.		R	A							
22												END
23												SUB 1
24				7.5								
25			-.300	F	A							
26	-1.250	-1.250		F	I							
27			.100	F	A							
28										Aux 800		
29												END
0												
1												
2												
3												
4.												
5												
6												
7												
8												
9												
0												

MIRROR IMAGE

B. EXAMPLE 7: MIRROR IMAGE

This example (Figure 42) has four slots .500 wide that go through the part. A .500 end mill will be used. Since the slots are the same width and length, the dimensions of the slot will be put incrementally into a subroutine and mirror image will be used to invert the direction that the slot moves.

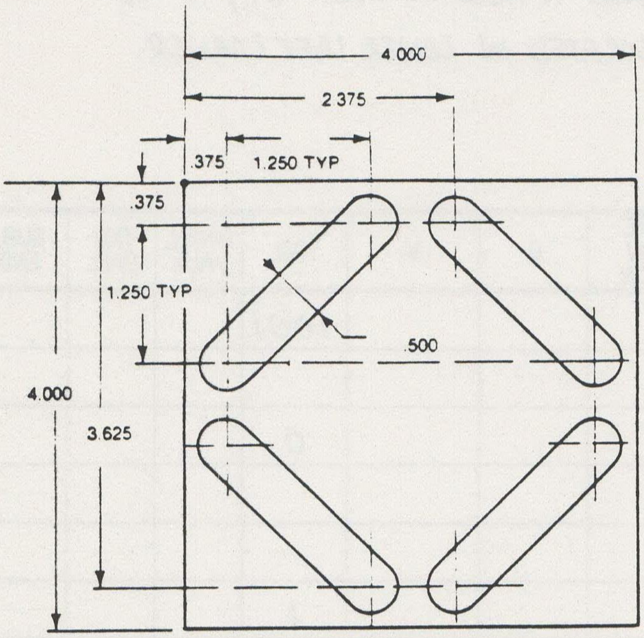


Figure 42: MIRROR IMAGE MILLING

In this subroutine, along with the moves to cut the slot, you will also put the moves that will feed the cutter into the workpiece. When the slot has been cut, this will also move the cutter up out of the piece. Finally, you will put in an AUX 800 to deactivate any other mirror image.

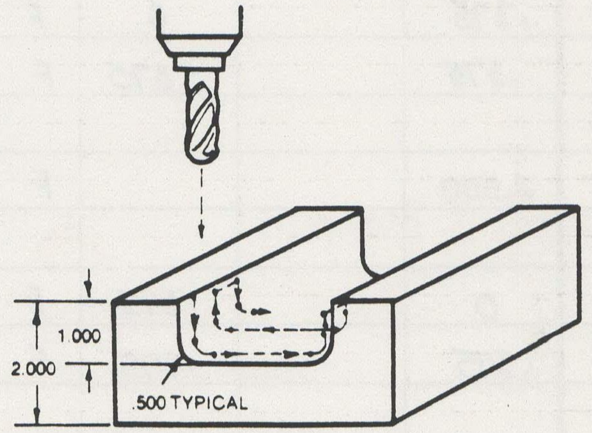
Refer to figure 6 at beginning of this manual for a diagram showing the mirror-image quadrants.

Event 1-6: These events have the same format as all the programs.
Event 7: Moves the cutter to the absolute position of the starting point for the first slot.
Event 8: Positions the cutter .100 above the part.
Event 9: Calls subroutine #1 to machine the slot.
Event 10.: Positions the cutter to the absolute position of the starting point for the next slot.
Event 11: Mirrors the X-axis dimensions.
Event 12: Calls subroutine #1.
Event 13: Positions to the start point of the next slot.
Event 14: Mirrors the X and Y-axis dimensions.
Event 15: Calls subroutine #1.
Event 16: Positions to the start point of the last slot.
Event 17: Mirrors the Y axis dimensions.
Event 18: Calls subroutine #1.
Event 19-22: Deactivates tool offset, retracts quill, moves to the tool change position and ends the program.
Event 23: Labels subroutine #1.
Event 24: Feedrate.
Event 25: Feeeds tool into the part.
Event 26: X and Y move to cut the slot.
Event 27: Feeds tool out of the part.
Event 28: AUX code to turn off all mirror images.
Event 29: Ends the subroutine.

C. EXAMPLE #8: X Z CONTOURING

The program for figure 43 uses a do loop to contour in the **X** and **Z** axis. Any contouring in the **XZ** or **YZ** axis must be done with a ballnose end mill or an end mill with a corner radius. Let us assume that the part has been roughed out and this program will take a finish cut.

The end mill will feed down along the edge of the part, contour the radius, feed to the center of the other radius, contour, feed up along the edge to the top of the part. The cutter will then increment .050 in the Y axis and do the entire sequence in reverse. This entire sequence is repeated 20 times to move a total distance of 2.000 inches; (2 x .050) x 20 = 2.000.



SHEET 1 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 8: CONTOURING PROGRAMMER WCS DATE 6/1/83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
TOOL 1: .250 DIAMETER BALLNOSE END MILL @ 3,500 RPM's		PART IS HELD IN VISE. X, Y "0" IS LOCATED IN LOWER LEFT CORNER.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2												
3									0			
4			0	R	A							
5	-2.000	2.000		R	A							
6									1			
7	.625	0		R	A							
8			.100	R	A							
9				F20.0								
10											DO 20	
11			-.625	F	A							
12						CW						
13	.375		0	F	I							
14	.375		-.375	F	I							
15						ARC						
16	2.500			F	A							
17						CW						
18	0		.375	F	I							
19	.375		.375	F	I							
20						ARC						

SHEET 2 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 8: CONTOURING PROGRAMMER WCS DATE 6/1/83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.		SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
21			-.125	F	A							
22		.050		F	I							
23			-.625	F	A							
24						CCW						
25	-.375		0	F	I							
26	-.375		-.375	F	I							
27						ARC						
28	1.000			F	A							
29						CCW						
30	0		.375	F	I							
31	-.375		.375	F	I							
32						ARC						
33			-.125	F	A							
34.		.050		F	I							
35												END
36									0			
37			0	R	A							
38	-2.000	2.000		R	A							
39												END
0												

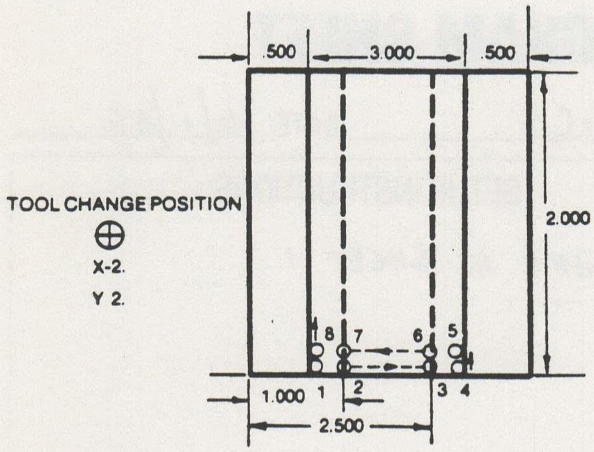


Figure 43: CONTOURING

Event 1-6:	Defines the tool offset, retracts the quill, moves to the tool change position and activates tool #1's offset.
Event 7:	Positions the center of the tool inside the left edge of the contour.
Event 8-11:	Feeds the tool to the start point of the first arc and starts the do-loop sequence.
Event 12-15:	Contours the first arc. Note that you are programming the **center** of the ball end mill, **not the bottom** of the cutter.
Event 16-20:	Feeds the tool to next arc start point and contours it.
Event 21-23:	Feeds the tool to the top of the part, increments .050 in the Y axis, then feeds down to the start point of the next arc.
Event 24-34:	Contours the arc, feeds the tool to the next arc and contours it. The tool then feeds to the top of the part and increments in Y .050.
Event 35:	Ends the do-loop which was started on event 10. In this way everything between events 10 and 35 will be repeated by using a do-loop.
Event 36-39:	Deactivates tool #1's offset, retracts the quill, rapids to the tool change position and ends the program.

SECTION 9 : ADDITIONAL CRUSADER FEATURES

A. CUTTER DIAMETER COMPENSATION

This feature should not be used until all functions of the control, discussed so far, are understood. It is strongly recommended that you use the control for at least one week before using cutter diameter compensation.

Previously, you have had to program the tool center. If a different size cutter is going to be used besides the one which was programmed, the program would have to be edited for the new tool size. Now, cutter diameter compensation can be used and the control will calculate the new dimensions for the new tool path. The program does not have to be completely re-edited.

Cutter diameter compensation can also be used to compensate for the full size of the cutter. By programming the part edge and using cutter diameter compensation, the new tool path at the center of the cutter can be calculated by the control. All of these situations will be covered in the following examples.

The default condition for cutter compensation is that it is active in the XY plane. If a ball-nose end mill is used, cutter compensation can be used in the YZ or XZ plane.

All of the following examples show only XY plane operations - but the same rules apply to XZ or YZ plane combinations.

To select the plane you wish to use cutter compensation in, program one of the following codes:

G17 - (defaults) selects XY plane.
G18 - selects XZ plane.
G19 - selects YZ plane.

Each of these three codes are modal and stay active until one of the other codes are programmed.

1. CALCULATING THE TOOL PATH.

A new tool path is calculated by the control to the left or right of the original programmed path. In Figures 44-45 you can assume the part edge was programmed. Therefore, the part edge is the original programmed path. In Figure 44, if the part was programmed from point A to point B and continued around the part in that direction, the cutter would have to be placed on the left of the original path to cut around the outside of the part. The command to compensate to the left is G41.

If the part was programmed from point B to point A and continued around the part in that direction (as in Figure 45), the cutter would have to be placed on the right of the original programmed path to cut around the outside of the part. The command to compensate to the right is G 42.

G41: CUTTER LEFT

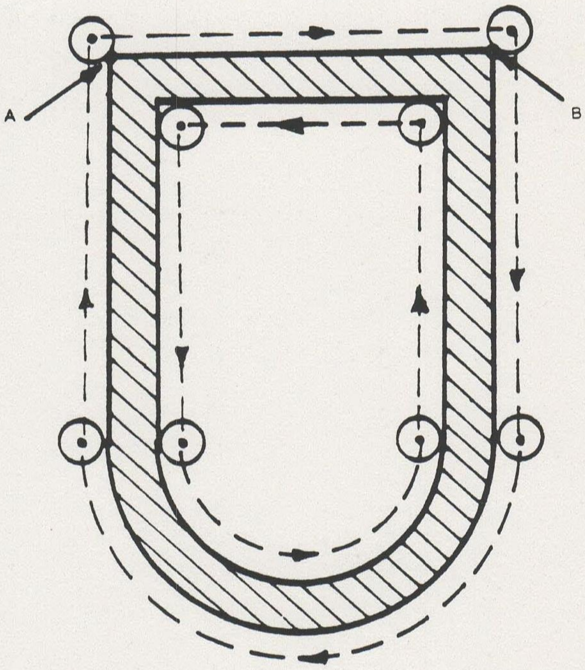


Figure 44

CAUTION:
When programming circles with cutter compensation, the programmed end point of the arc must be on the arc. If not and the next move is incremental, a position error will result.

G 42: CUTTER RIGHT

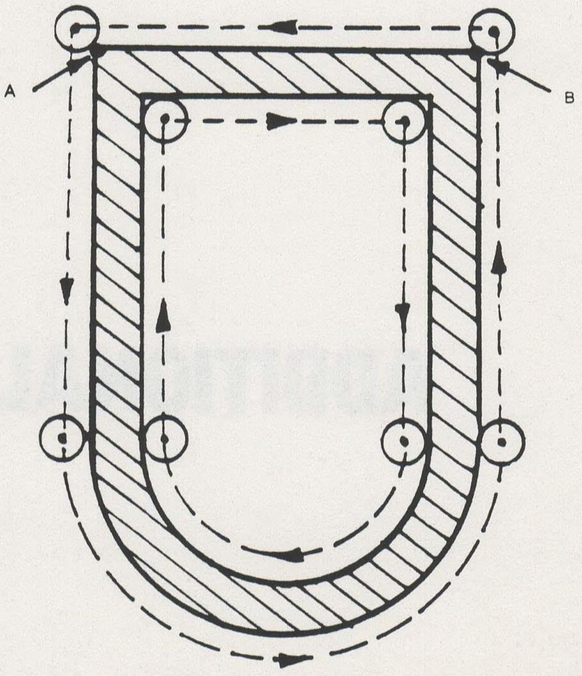


Figure 45

Study figures 46 and 47. If the part edge were programmed, write down the correct command (G 41 or G 42) to move the cutter as shown from position A to B. The answers are below. Also write down the command (G 41 or G 42) to move the cutter as shown from B to A.

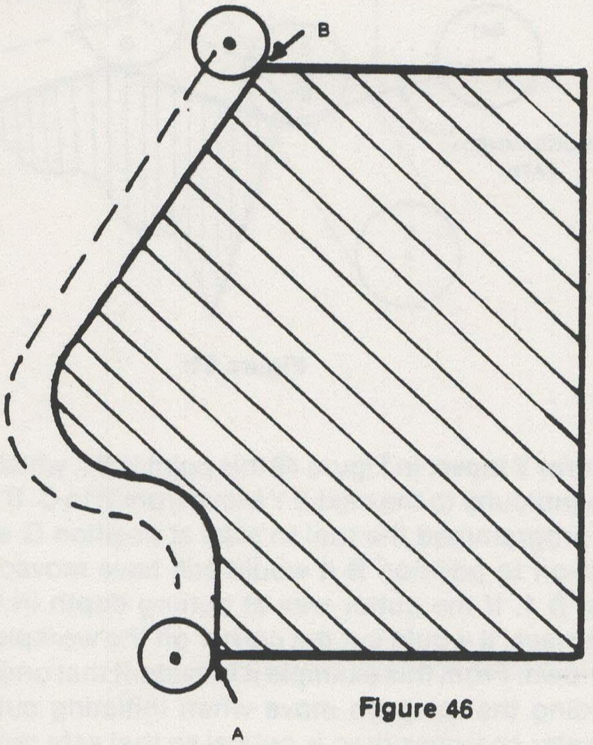


Figure 46

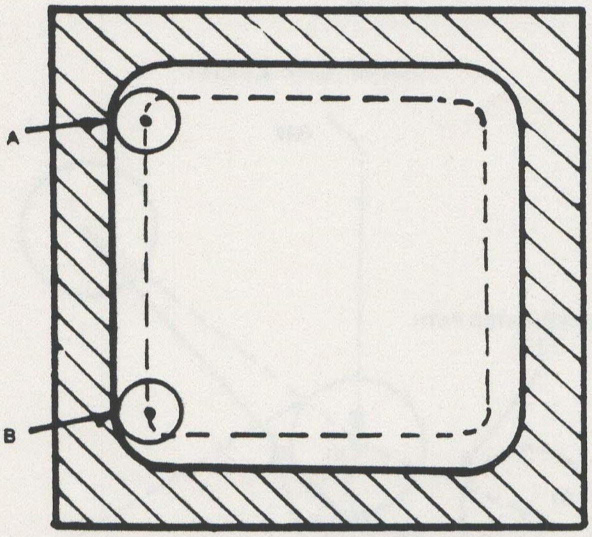


Figure 47

Figure 46: From A to B: G 41
From B to A: G 42

Figure 47: From A to B: G 41
From B to A: G 42.

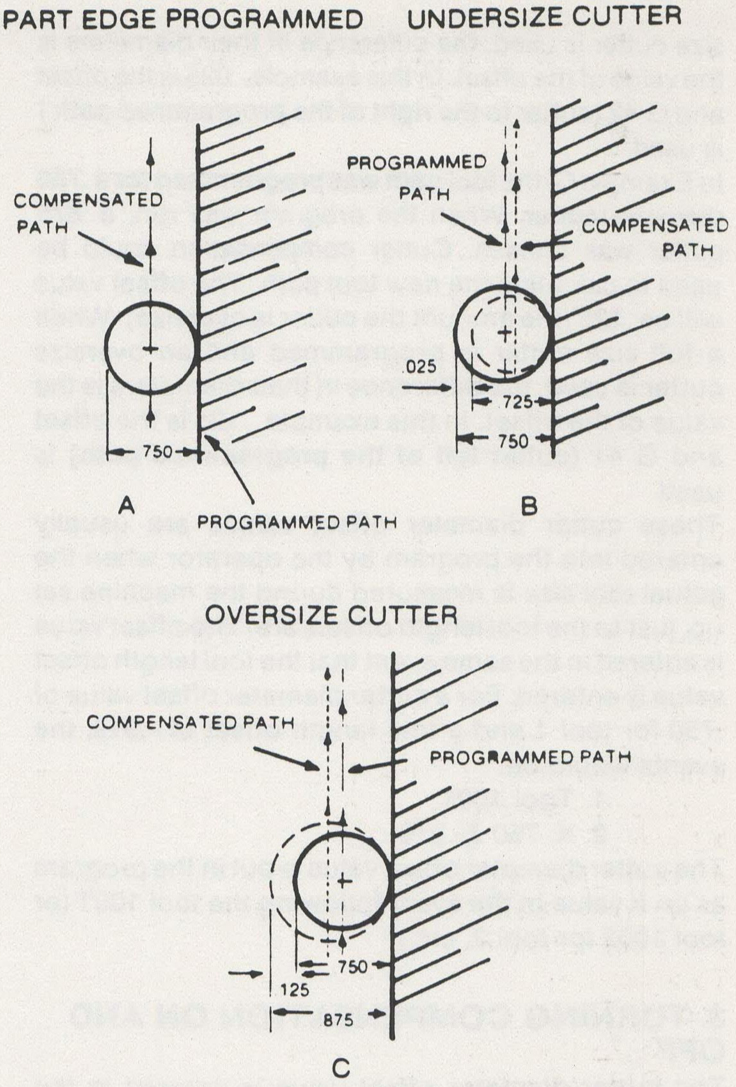


Figure 48

2. DISTANCE OF COMPENSATION

The distance the cutter is to stay away from the programmed path will depend on the size of the cutter being used and the way the part is programmed. In Figure 48A, the **part edge was programmed** and cutter compensation is used for the full diameter of the cutter. The value of compensation in this case is .750 (the full diameter of the cutter). When programming the part edge, the diameter of the cutter being used will be the value of the offset. In this example, .750 is the offset and G 41 (cutter left) is used.

In Example B, the tool path **at the center** of a .750 cutter was programmed. When the program was run, a .725 cutter was all that was available. Cutter compensation could be used to calculate the new tool path. The offset value will be .025 (the difference between cutter sizes). When a full size cutter is programmed and an under-size cutter is used, the difference in their diameters is the value of the offset. In this example, .025 is the offset and G 42 (cutter **to the right of the programmed path**) is used.

In Example C, the **tool path was programmed for a .750 diameter cutter.** When the program was run, a .875 cutter was chosen. Cutter compensation could be used to calculate the new tool path. The offset value will be .125 (the amount the cutter is oversize). When a full size cutter is programmed and an oversize cutter is used, the difference in their diameters is the value of the offset. In this example, .125 is the offset and G 41 **(cutter left of the programmed path)** is used.

These cutter diameter offset values are usually entered into the program by the operator when the actual tool size is measured during the machine set up, just as the tool length offsets are. The offset value is entered in the same event that the tool length offset value is entered. For a cutter diameter offset value of .750 for tool 1 and a tool length offset of -.919, the events would be:

1. Tool 1001
2. X. 750 Z-.919

The cutter diameter offset value is put in the program as an X value in the event following the tool 1001 (or tool 1002 for tool 2, etc.)

3. TURNING COMPENSATION ON AND OFF

The cutter diameter offset value is entered in the program after **TOOL 1001** . This offset must then be activated later by programming **TOOL 1.** The TOOL 1 event will tell the control to use the offsets described for TOOL 1.

If no offsets were given in the event following **TOOL 1001,** the control will assume an offset of zero. After the tool is programmed, the control will make the offsets active. The tool length offset will become active immediately. A G41 or G42 must be programmed so the control knows if compensation is to take place on the left or on the right. **The G 41 or G 42 event activates the cutter diameter offset during the next X and/or Y move.** This will be called the "ramp-on" move.

In Figure 49, an offset value is given for tool 1 and is activated by programming **TOOL 1.** The tool is programmed to move to position A and a G 41 event is entered. During the next X, Y move to position B, the cutter diameter compensation will take effect.

After a G 41 or G 42, the center of the cutter will always ramp-on to a point perpendicular to the next X and/or Y move. In Figure 49 this point is B1, which is perpendicular to the next X Y Move from B to C. If we had programmed the tool to start at position G and ramp-on to position B it would still have moved to point B 1. If the cutter was at cutting depth in the workpiece, it would cut the corner off the workpiece as shown. From this example it is evident that understanding the ramp-on move when initiating cutter diameter compensation is critical so that safe ramp-on moves will be programmed.

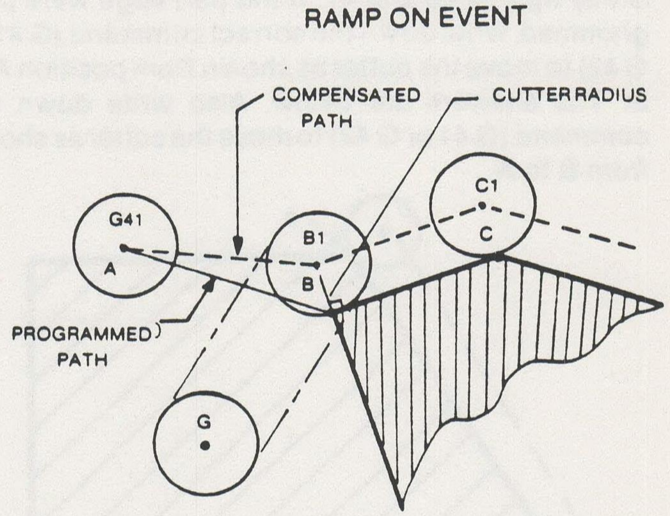


Figure 49:

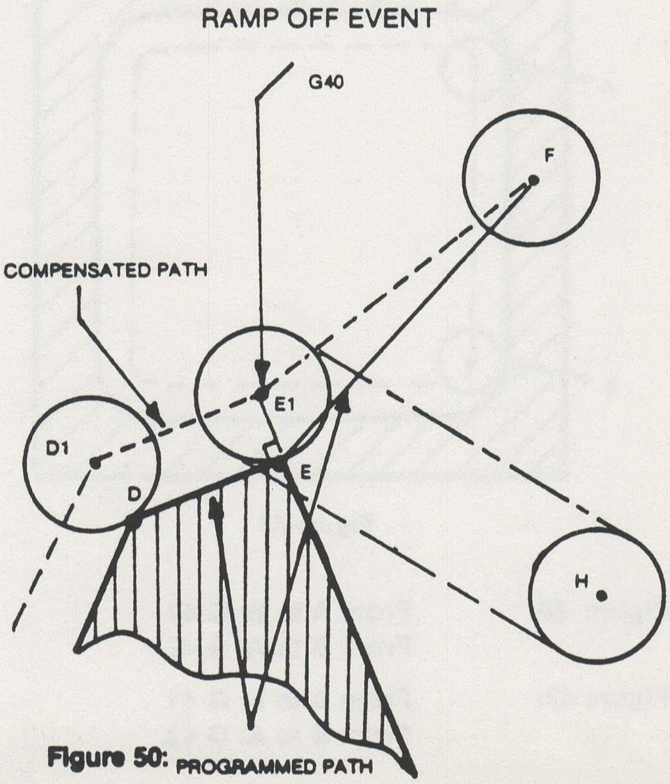


Figure 50:

Figure 50 shows what happens when cutter compensation is turned off. G 40 is used to turn off compensation. After the event to move the tool to position E was entered, a G 40 was entered. A G 40 causes the cutter to stop at a position (E 1) **which is perpendicular to the previous X and/or Y move.** The previous X Y move was from D to E, so the tool will stop at E 1.

After the G 40 event, the cutter will ramp-off during the next Z and/or Y move. This will be called the ramp-off move. Notice the path the tool would take from point E 1 to point F. If programmed from point E to point H with the cutter at cutting depth, it would cut the corner off the workpiece.

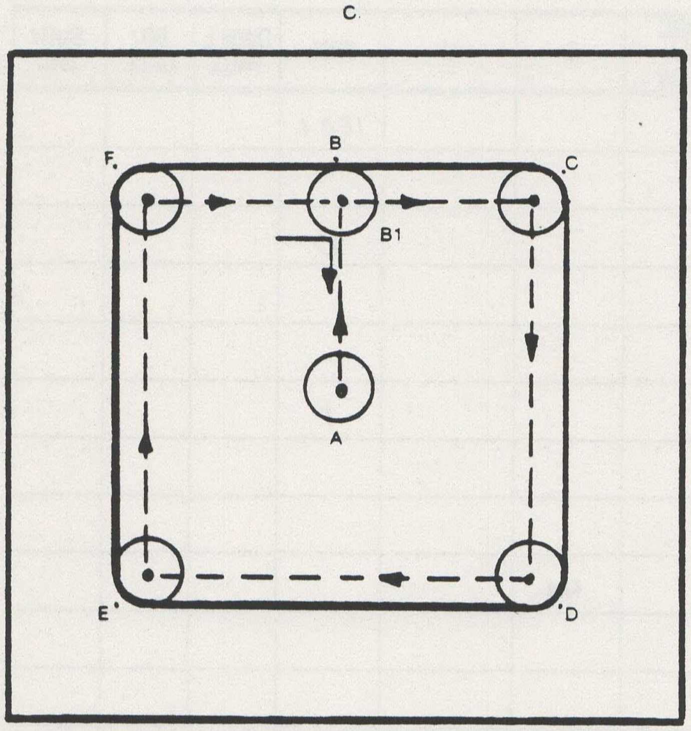


Figure 51

COMPENSATING AN INSIDE POCKET

Figure 51 shows the correct way to program an inside pocket. Point B must be at least one cutter radius away from the corner of the pocket. The pocket was programmed: point A, G 42, point B, C, D, E, F, B, G 40 and back to A. Following the rules above, the ramp-on move is the Y-axis tool motion from A to B. The tool stops at B 1 which is perpendicular to B C. B C is the next X-axis move. (The tool will also stop at B 1 when G 40 is programmed after the move from F to B.)

Figure 52 shows what would happen if you programmed the ramp-on move from A to C. Following the rules, the tool would move to C 1 which is perpendicular to the next Y-axis move to C to D. A similar problem would occur if a G 40 was programmed after moving from F to C. The tool would stop at C2, perpendicular to the move F to C.

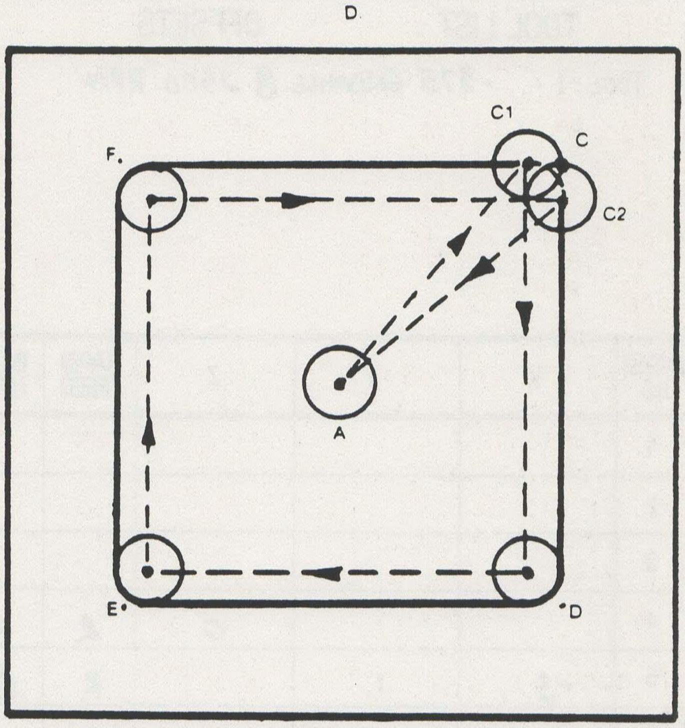


Figure 52

The same part which was drilled in Example 1 will be used to demonstrate cutter diameter compensation. See Figure 53. The part edge will be programmed from A to B, C, D, E, F, G, H. The full diameter of the cutter (.375) will be the offset value and cutter diameter compensation will be turned on between point A and B causing the tool to go from point A to B1, C1, D1, E1, F1, G1. Compensation will be turned off between G and H which will cause the tool to move to point H. Use the part drilled in Example 1 and cut this shape into the part using cutter diameter compensation and an available 2 flute end mill. Enter its diameter and length offsets in Event 2.

Event 1:	Tool 1001 is tool 1 offset definition
Event 2:	Left open for operator to enter offset values.
Event 3:	Deactivates offsets.
Event 4:	Retracts quill.
Event 5:	Rapids to tool change position.
Event 6:	Activates tool 1's offsets.
Event 7:	Rapids tool to position A.
Event 8:	Rapids tool to cutting depth.

SHEET 1 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 1 w/CUTTER COMP. PROGRAMMER RJT DATE 4 · 18 · 83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
TOOL 1: .375 ENDMILL @ 2300 RPM		XY "0" AT UPPER LEFT CORNER. ACTUAL MEASURED DIAMETER OF TOOL USED ENTERED AS "X" VALUE IN EVENT 2 WITH Z-AXIS TOOL LENGTH OFFSET.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2												
3									0			
4			0	R	A							
5	-3.	1.		R	A							
6									1			
7	6.	-1.1875		R	A							
8			-.35	R	A							
9							41					
10				F4.								
11	5.	-1.		F	A							
12	4.	-1.5		F	A							
13	2.5			F	A							
14.				F2.5								
15						ARC CCW						
16	2.5	-2.		F	A							
17	2.5	-2.5		F	A							
18						ARC						
19				F4.								
20	4.			F	A							

ANILAM **Anilam Electronics Corporation** 5625 N.W. 79th Avenue, Miami, Florida 33166 / (305) 592-2727 / Telex 26-4024

SHEET 2 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 1 w/CUTTER COMP. PROGRAMMER RJT DATE 4·18·83

TOOL LIST OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET #1.	SAME AS SHEET #1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
21	5.	-3.		F	A							
22							G40					
23	6.	-2.8125		F	A							
24									0			
25			0	R	A							
26	-3.	1.		R	A							
27												END
8												
9												
0												
1												
2												
3												
4.												
5												
6												
7												
8												
9												
0												

ANILAM **Anilam Electronics Corporation** 5625 N.W. 79th Avenue, Miami, Florida 33166 / (305) 592-2727 / Telex 26-4024

Event 9:	Activates cutter compensation to the left.
Event 10:	Sets the milling feedrate.
Event 11:	Causes the tool to move to point B1 because G 41 is active. This is the ramp on event.
Event 12:	Tool feeds to point C1.
Event 13:	Tool feeds to point D1.
Event 14:	Adjusts feedrate for constant surface speed.
Event 15:	Starts arc counter clockwise.
Event 16:	Defines arc center.
Event 17:	Defines arc finish point.
Event 18:	Arc end command.
Event 19:	Sets feedrate back to 4 IPM
Event 20:	Tool feeds to point F1.
Event 21:	Feeds tool to point G1 because a G 40 is programmed before the next X Y move.
Event 22:	Deactivates cutter diameter compensation.
Event 23:	Tool feeds to point H. This is the ramp-off event.
Event 24:	Deactivates tool offsets.
Event 25:	Retracts quill.
Event 26:	Rapid to tool change position.
Event 27:	End of program.

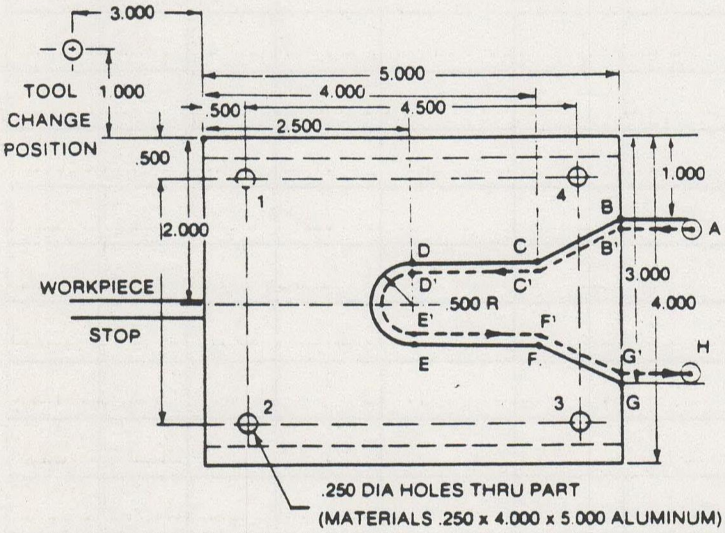


Figure 53: CUTTER COMPENSATION (Example 1)

4. EXAMPLE 9: CUTTER COMPENSATION

Figure 54 shows a hex-shaped part which will be programmed using cutter compensation. The part edge will be programmed and the full diameter of the tool will be entered as the offset value. The part will be held in a fixture and absolute zero will be set in the center of the part. The speeds and feeds will be set for a .375 diameter cutter but actual cutter diameters can be entered and used with cutter diameter compensation. The actual cutter used should be as close to .375 as possible to use the programmed feeds and speeds.

The tool change position will be at X-2.000 and Y 2.000 absolute. The tool will be programmed to move to points A, B. C, D, E, F and back to A using cutter compensation. G 41 will be turned on at point 1. As the previous rules in section 3 state, the cutter will ramp-on during the next X Y move to point A and stop at point 2 which is perpendicular to the next X Y move from A to B. The tool will stay at the compensated distance away from the part around points 3, 4, 5, 6, 7 and 8. The program will be the dimensions of points A through F and back to A.

EXAMPLE 9

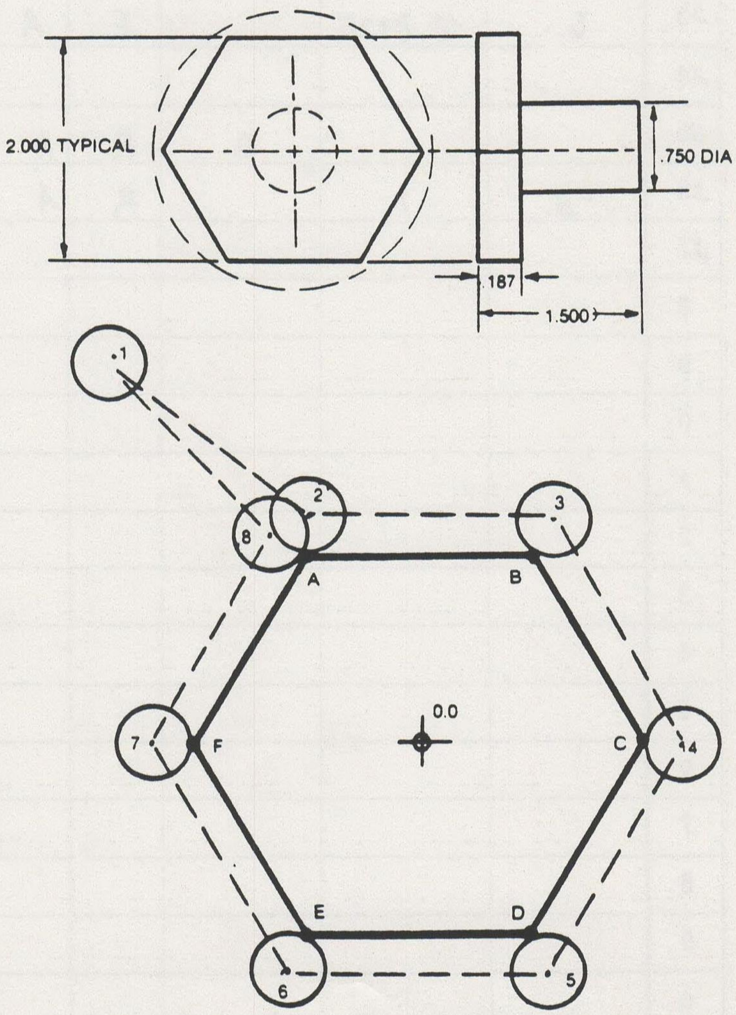


Figure 54

At point A, G 40 is programmed. This will cause the cutter to stop at point 8. Again this follows the rules of

SHEET 1 OF 1

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 9: CUTTER COMP. PROGRAMMER RPS DATE 4·1·83

TOOL LIST OFFSETS	SETUP INSTRUCTIONS
TOOL 1: .375 ENDMILL @ 1,500 RPM's	PART HELD IN FIXTURE. X,Y "0" SET AT PART CENTER. CUTTER DIAMETER OFFSET ENTERED IN EVENT 2 AS FULL DIAMETER OF TOOL.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2												
3									0			
4			0	R	A							
5	-2.	2.		R	A							
6									1			
7			-.4	R	A							
8							41					
9				F5.0								
10	-.5774	1.		F	A							
11	.5774			F	A							
12	1.1547	0		F	A							
13	.5774	-1.		F	A							
14.	-.5774			F	A							
15	-1.1547	0		F	A							
16	-.5774	1.		F	A							
17							40					
18	-2.	2.		R	A							
19									0			
20			0	R	A							
21												END

SHEET 1 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 10: POCKET CUTTER COMP. PROGRAMMER RCH DATE 11.25.83

TOOL LIST OFFSETS	SETUP INSTRUCTIONS
TOOL 1: .250 END MILL @ 3,000 RPM's	PART HELD IN VISE. X,Y "0" LOCATED IN LOWER LEFT CORNER.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1									1001			
2	.020											
3									0			
4			0	R	A							
5	-2.	1.		R	A							
6									1			
7	2.	1.75		R	A							
8			-.3	R	A							
9				F 10.0								
10											CALL 1	
11									1001			
12	0											
13									1			
14.				F20.0								
15											CALL 2	
16									0			
17			0	R	A							
18	-2.	1.		R	A							
19												END
20												SUB 1

SHEET 2 OF 2

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # EXAMPLE 10: POCKET CUTTER COMP. PROGRAMMER RCH DATE 11.25.83

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS
SAME AS SHEET 1.		SAME AS SHEET 1.

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
21							41					
22		2.25		F	A							
23	1.			F	A							
24						ARC CCW						
25	0	-.375		F	I							
26	-.375	-.375		F	I							
27						ARC						
28		-1.		F	I							
29						ARC CCW						
30	.375	0		F	I							
31	.375	-.375		F	I							
32						ARC						
33	1.5			F	I							
34						ARC CCW						
35	0	.875		F	I							
36	0	1.75		F	I							
37						ARC						
38	2.			F	A							
39							40					
40		1.75		F	A							
41												END

turning compensation off. Point 8 is perpendicular to the previous X Y move from F to A. The tool ramps-off after G 40 during the next X Y move from 8 to 1. Refer to Figure 54 as needed during programming.

Event 1:	TOOL 1001 defines tool 1's offsets.
Event 2:	Left open for entering the cutter diameter offset value in X and the tool length offset in Z.
Event 3-6:	Retracts the quill, moves to the tool change position and puts Tool 1's offsets in effect.
Event 7:	Moves the tool -.400 below the top of the part.
Event 8:	Sets compensation to the left.
Event 9:	Sets the feedrate.
Event 10:	The ramp-on move in which the tool will move to position 2.
Event 11-16:	The remaining coordinates calculated for points B through A.
Event 17:	Deactivates cutter diameter compensation (not tool length compensation). The tool will stop at point 8.
Event 18:	The ramp off move back to X-2. Y 2.
Event 19-21:	Retracts the quill and ends the program.

Figures 55-57 show how the tool path is adjusted when using G 41 and G 42. The solid lines represent the programmed path, the dashed lines represents the tool path.

5. RULES OF CUTTER DIAMETER COMPENSATION

These rules apply **only** when using cutter diameter compensation:

1. The offset value is given as an X axis dimension after the TOOL 1001 (or TOOL 1002 etc.) definition event.
2. Tool 1 (or Tool 2, etc.) must be programmed before G 41 or G 42 so the control will know which tool number's offset is active.
3. The cutter ramps-on during the next X and/or Y move **after** a G 41 or G 42.
4. The cutter ramps-on perpendicular to the next X and/or Y move.
5. There must be at least one straight line move after G 41 or G 42 before cutting an arc. (See figure 55).
6. An arc can be only be followed by a straight line move or another arc. G 40 **is not permitted** immediately following an arc. (See figure 57).
7. Z-axis moves are permitted at any time.
8. Changing tool numbers (including changing to Tool O or ending the program) is not permitted until after the ramp-off move following G 40.
9. G 40 causes the tool to stop perpendicular to the last X and/or Y move.
10. Each compensated move must have an intersection.

Figure 55

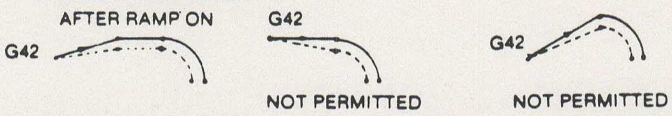


Figure 56

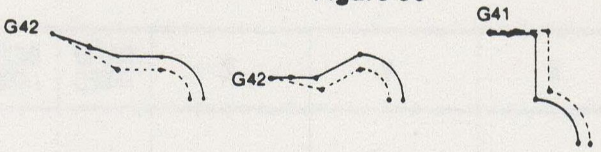
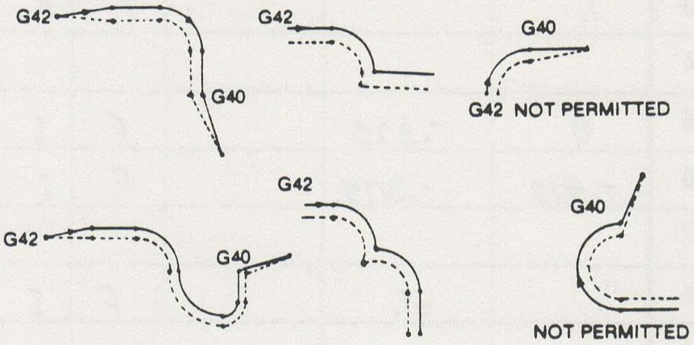


Figure 57:



6. EXAMPLE 10: POCKET CUTTER COMPENSATION.

The part in figure 58 has a window which is roughly cast. A rough cut and finish cut is required to finish the part. A .250 diameter end mill will be used. The tool path at the **center of the cutter** will be programmed in a sub-routine. The cutter will be defined as having a .020 cutter diameter offset for the rough cut. This will leave .010 on a side for the finish cut. With .020 offset active, the sub-routine will be called to rough out the window. The offset value will be changed to zero and the same routine will be called to finish the window. In this way the same tool and subroutine can be used with different amounts of compensation to rough and finish a part.

To illustrate this, the tool path is plotted out in the bottom drawing in figure 58. The tool path in subroutine 1 will be points 2, 3, 4, 5, 6, 7, 8, 9 back to 3 then 2.

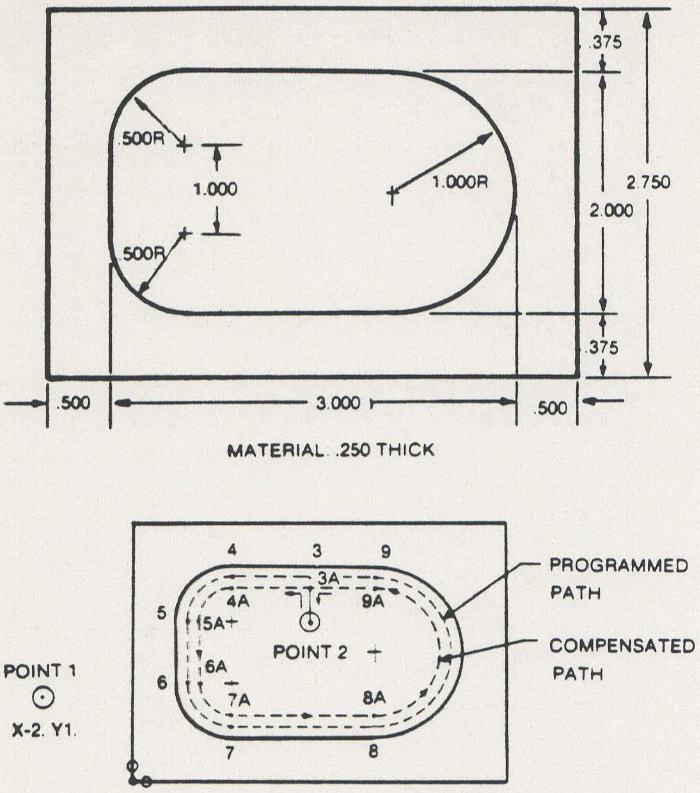


Figure 58: POCKET CUTTER COMPENSATION

A G 41 will be programmed between 2 and 3 causing the cutter to ramp-on and stop at point 3A for the rough cut. Compensation will take place all the way around the part. After point 3A is reached after cutting the window, a G 40 will be entered and the tool will ramp-off back to point 2.

Compensation can be used because the cutter does not ramp-off immediately following an arc.

Event 1:	Tool 1001 to define tool 1's offsets.
Event 2:	X .020 sets the offset to rough cut the part with .010 material left for a finish cut. The tool length offset will be entered during set up.
Event 3-6:	Retracts the quill, moves to the tool change position and activates tool 1's offsets.
Event 7:	Rapids the tool to position 2.
Event 8:	Rapids the tool to cutting depth. The window is cast in the part and the tool rapids through the window.
Event 9:	Sets the feedrate for roughing at 10.0 IPM.
Event 10:	Calls subroutine 1 which roughs the window with .020 cutter diameter compensation. The subroutine initiates G 41, ramps on, contours around the window, cancels using G 40 and ramps off, back to point 2.
Event 11:	Redefines tool 1's offsets.
Event 12:	Sets the tool diameter compensation to zero to finish the window. The tool length offset must be reentered during set up in this event also.
Event 13:	Activates the new offset for tool 1. Because the same tool number is programmed twice consecutively, the program will not stop.
Event 14:	Sets the feedrate to 20 ipm for the finished cut.
Event 15:	Executes the subroutine with zero diameter offset for a finish cut.
Event 16-19:	Retracts the quill and ends the program.
Event 20:	Starts subroutine 1's definition.
Event 21:	Activates G 41, cutter left.
Event 22:	Ramp on move to point 3 or 3A depending on which offset is active. With zero offset active, the cutter will move to point 3. With .020 offset active, the cutter will move to point 3A.
Event 23-38:	Moves the cutter around the window in the part. The offset which is active will determine which points the tool moves to.
Event 39:	Deactivates G 41 with G 40. This causes the tool to stop perpendicular to the move from 9 to 3 and stop at 3A. With zero offset in effect the tool would stop at point 3.
Event 40:	The ramp off move to point 2.
Event 41:	Ends the subroutine definition.

SECTION 10: MISCELLANEOUS

A. AUXILIARY CODE LISTING

The AUX button is used in the main program to cause variations in the standard control functions. These variations are assigned numbers and are entered into the program prefixed by the AUX button (i.e. AUX 7, AUX 400, AUX 2500, etc.). Below is a list of all the AUX codes and functions in both the standard Crusader control and the optional Advanced Software System-1 package.

AUX codes 100-800 affect the Mirror Image feature of the Crusader control. A mirror image is usually referred to as a left-hand or right-hand part. These mirror image AUX codes will reverse the plus or minus signs of a programmed dimension to produce an "opposite" part.

A mirror image will stay active until deactivated by AUX 800 (see list) or another mirror image. Using EVENT SEARCH or changing modes (except changing from SINGLE STEP to AUTO) will also deactivate mirror image. For examples see Examples 5 and 7 in this manual. The hole patterns and milled shapes in figures 59, 60 and 61 could be programmed using mirror image.

NOTE:

All mirror image AUX codes stay active until cancelled.

AUXILIARY MIRROR IMAGE

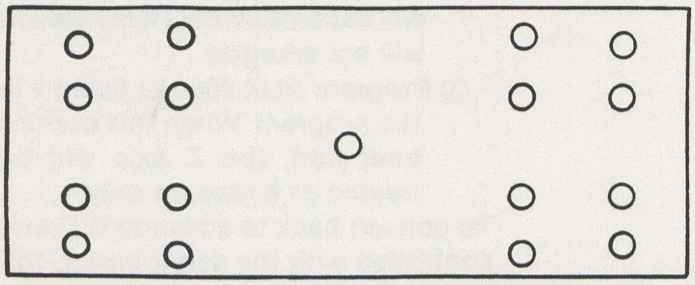


Figure 59

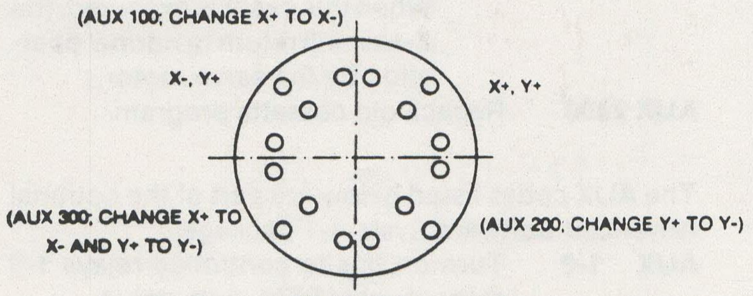

(AUX 100; CHANGE X+ TO X-)
X-, Y+
X+, Y+
(AUX 300; CHANGE X+ TO X- AND Y+ TO Y-)
(AUX 200; CHANGE Y+ TO Y-)
X-, Y-
X+, Y-

Figure 60

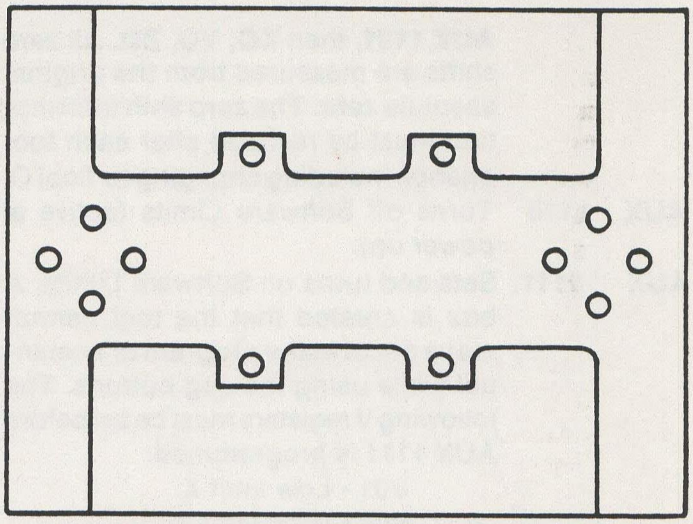


Figure 61 AUXILIARY MIRROR IMAGE

AUX	**100**	Reverses the signs of the X axis dimensions.
	200	Reverses the signs of the Y axis dimensions
	300	Reverses the signs of the X and Y axis dimensions.
	400	Reverses the signs of the Z axis dimensions.
	500	Reverses the signs of the X and Z axis dimensions.
	600	Reverses the signs of the Y and Z axis dimension.
	700	Reverses the signs of the X, Y and Z axis dimensions.
	800	Turns the mirror image off.

Other standard AUX codes are as follows:

AUX	**1000**	Activate Continuous Path. This should only be used for contouring continous tangent lines and circles and should be turned off immediately after use. It is not to be used in Rapid or when doing consecutive straight line moves or over 40 inches per minute in Feed. This command is modal and becomes active for the next motion command. It causes pre-processing to take place, so the machine doesn't pause between moves but executes continous motion.
AUX	**1101**	(Absolute Zero Shift) AUX 1101 followed by X 6., Y 1., Z-.903 will shift absolute zero to these X, Y and Z coordinates. To shift the zero back to the original absolute zero, program

AUX 1101, then XO, YO, ZO. All zero shifts are measured from the original absolute zero. The zero shift information must be restated after each tool change including changing to Tool O.

AUX 1110 Turns off Software Limits (active at power up).

AUX 1111 Sets and turns on Software Limits. A box is created that the tool cannot move out of in the program or in manual while using the Jog buttons. The following V registers must be set before AUX 1111 is programmed.

V01 - Low limit X.
V02 - High limit X.
V03 - Low limit Y.
V04 - High limit Y.
V05 - Low limit Z.
V06 - High limit Z.

The tool must be inside the box created when AUX 1111 is activated. The software limits are not moved by shifting absolute zero.

AUX 1112 Reactivates Software limits previously established by AUX 1111. Allows the operator to activate and deactivate software limits without constant reprogramming.

AUX 1400 Feed% override for feed moves only. Rapid moves will be made at 100 IPM. (When the control is turned on, this function is automatically active each time.)

AUX 1401 Feed% override for Feed and Rapid. Both Feed and Rapid moves will be affected by Feed% override.

AUX 1410 Causes rapid motion to be executed in a straight line. This is a modal command and is active at power up.

AUX 1411 Allows all axis to run at maximum rapid speed when a rapid move is programmed. The tool will not travel in a straight line when this AUX-code is active. This is a modal command.

AUX 1160 Deactivate Backlash compensation (default).

AUX 1161 Set and activate Backlash compensation. The following variables must be set before this AUX-code is activated:

V01 - distance of backlash compensation in the X- direction.
V02 - same in the X+ direction
V03 - same in the Y- direction
V04 - same in the Y+ direction
V05 - same in the Z- direction
V06 - same in the Z+ direction

AUX 1162 Reactivate Backlash compensation using variables established in AUX 1161.

AUX 1900 SINGLE STEP mode will execute one event each time the START button is pressed.

AUX 1901 SINGLE STEP mode will execute one X, Y or Z move each time the START button is pressed (active at power up).

AUX 2000 Cancels AUX 1000 (active at power up).

AUX 2500 Cancels 2600. Sets the Control to read and execute Z-axis commands (active at power up).

AUX 2600 Cancels AUX 2500. Sets the Control to disregard Z-axis motion so the Z-axis can be used manually with handle while the X and Y axis are controlled by the servo motors. To convert the Z-axis from a servo controlled axis to a manually controlled axis with readout, do the following steps:

1) Press EMERGENCY STOP.
2) Unplug the Z-axis servo motor cable from the cabinet. Replace with jumper plug if required. This will become evident if the servos will not energize.
3) Program AUX 2600 as Event 1 in the program. When this event is executed, the Z axis will be treated as a readout only.

To convert back to a standard Z-axis controlled with the servo motor, follow these steps:

1) Press EMERGENCY STOP.
2) Reconnect Z-axis motor cable.
3) Program AUX 2500 as Event 1. When this event is executed, the Z-axis will return to normal operation by the servo motor.

AUX 2800 Repack old cassette program.

The AUX codes listed below are part of the optional Advanced Software System-1 package:

AUX 1-8 Turn on bits to controlled relays 1-8 respectively (OEM units only.)

AUX 9 Turn off bits to controlled relays.

AUX	**11-18**	Turn off bits to individual relays 1-8 respectively (OEM units only.)

The following AUX 2700 series are active with RS-232-C option only.

AUX	**2700**	Write to RS—232-C device in RS 274 format
AUX	**2701**	Read from RS-232-C device in RS 274 format
AUX	**2702**	Write to RS-232-C device in Anilam format
AUX	**2765**	Set 5 bits per character
AUX	**2766**	Set 6 bits per character
AUX	**2767**	Set 7 bits per character
AUX	**2768**	Set 8 bits per character
AUX	**2770**	Set to no parity
AUX	**2771**	Set to odd parity
AUX	**2772**	Set to even parity
AUX	**2780**	Set baud rate to 110 bits/second
AUX	**2781**	Set baud rate to 150 bits/second
AUX	**2782**	Set baud rate to 300 bits/second
AUX	**2783**	Set baud rate to 600 bits/second
AUX	**2784**	Set baud rate to 1200 bits/second
AUX	**2785**	Set baud rate to 1800 bits/second
AUX	**2786**	Set baud rate to 2400 bits/second
AUX	**2787**	Set baud rate to 4800 bits/second
AUX	**2788**	Set baud rate to 9600 bits/second
AUX	**2789**	Set baud rate to 19200 bits/second

Any speed higher than 1200 Bits/Second will transfer data in bursts. (Must use handshake).

AUX	**2790**	Set no handshake
AUX	**2791**	Set software handshake (X on, X off)
AUX	**2792**	Set hardware handshake (DTR, DST)

B. TAPE CASSETTE

The tape cassette is used to store a program which has been entered in the Crusader's memory so that you can repeat the same program at a later time. Record a program only after it has been proven and when there will be no more changes.

To record a program on tape from the Crusader memory:

1. Plug the cassette recorder unit into the rear of the console.
2. Press PROGRAM ENTER
3. Press **EMERGENCY STOP.**
4. Put a cassette tape in the recorder and close the spring-loaded flip-out door.
5. Press **RECORD.**

At this time, there may be a slight pause then the record light will come on. The tape will rewind, then the MANUAL light will come on. The PROGRAM CHECK light will blink meaning Channel 1 is being recorded on, and will stop blinking when recording is done. It will blink again when recording Channel 2. The tape will then rewind again.

The PROGRAM ENTER light will blink meaning that Channels 1 and 2 are being compared to the memory. The tape will rewind again and the MANUAL light will shut off. A tone will sound to signal you that the program has been recorded.

There are 2 sides to each tape and a different program can be recorded on each side. Identify the tape by writing the part number or job number from the program sheet on the tape.

To play a tape back into the Crusader's memory:

1. Plug the cassette player unit into the rear of the console.
2. Press **PROGRAM** and then press **EVENT CLEAR** 5 times to clear memory.
3. Press **EMERGENCY STOP.**
4. Put the cassette tape in the recorder and close the spring-loaded flip-out door.
5. Press **PLAY.**

At this time there may be a slight pause and the PLAY light will come on. The tape will rewind. The MANUAL light will come on. The PROGRAM ENTER light will blink while the memory is being loaded. The tape will then rewind again. The MANUAL light will then shut off and a tone will sound to signal that the tape has been played into the memory (disregard the displayed event number). Press PROGRAM CHECK, 1, EVENT SEARCH to verify that the program has been played back properly. If an error code is displayed, refer to the Error Code Listing.

IMPORTANT

The standard (and optional) software changes made in February 1983 with the Anilam Crusader has resulted in changes in the tape cassettes as well.

Tapes which were made with software manufactured **before** February 1983 will play into the new software units (made **after** February 1983) but there will be gaps between events. Immediately after playing old tapes into a new software console, pull out the Emergency Stop button, press **MANUAL AUX 2800 START.** This will eliminate the gaps by compressing the program.

Tapes made with software manufactured **after** February 1983 will not play into units with older software.

C. ERROR LIGHT

When certain errors are made, a tone will sound, the ERROR light will come on, and an error number will be displayed in the upper right display. To demonstrate this, press **PROGRAM ENTER** and press **RECORD.** Do **not** put a tape in the cassette recorder unit. The ERROR light will come on and the number 3 will be displayed. Refer to the list below for the error code and explanation.

ERROR	EXPLANATION
1.	Too many digits entered through the keyboard. The control will only accept dimensions from -999.9999 to -.0001 inches (-9999.99 to .01 mm) in the axis display registers and a maximum of a 4 digit number is allowed in the top 2 displays.
2.	No cassette recorder power. The cassette unit does not have power to run the tape winding mechanism.
3.	Cassette tape not in place. No tape in cassette recorder unit.
4.	Program cannot be recorded because safety tab is missing. After a tape has been proven, by removing this tab, the tape cannot be erased or changed.
5.	No program in memory to record.
6 & 7	The tape has not played correctly. Look at the program in PROGRAM CHECK and replay the tape if it is not correct.
9.	No program on the cassette you are trying to play or cassette has stopped.
10.	The program did not record on the cassette tape.
11.	Circle coordinates entered incorrectly. Either all three axis were described or the arc statement was left out after the end point.
12.	Software limit reached.
13.	Compensated elements do not intersect when cutter compensation is active.
15.	Z- speed command too fast.
16.	Acceleration too high.
17.	Opposing distance and velocity commands.
18.	No motion seen by control when command given. Probable causes are: A. Scale cable disconnected. B. Scale needs cleaning. C. D/A signal disconnected. D. Servo Amp fuse blown. E. Motor disconnected. F. Pulley or belt not functioning. G. Table lock on or gib seized. H. Ballscrew inoperable.
19.	Following error increasing too fast.
20.	Velocity command too large.
30.	Non-existent AUX 2700 code.
31.	User abort of RS 232 data transfer by pressing EMERGENCY STOP
32.	Program memory overflow while reading RS 232 DATA.
33.	Input error while reading RS 232 DATA
34.	Input error while outputting the program using X on, X off, using RS-232
35.	Buffer overflow using RS-232.

D. PRESETTING ZERO

Figure 62 shows how to use the preset capability in the Crusader to set zero. Rather than jog to zero after touching the edge of the part or using an indicator, simply tell the control where the center of the spindle is relative to absolute zero.

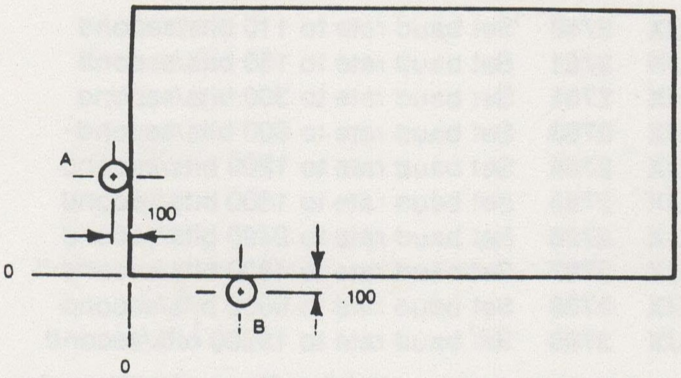
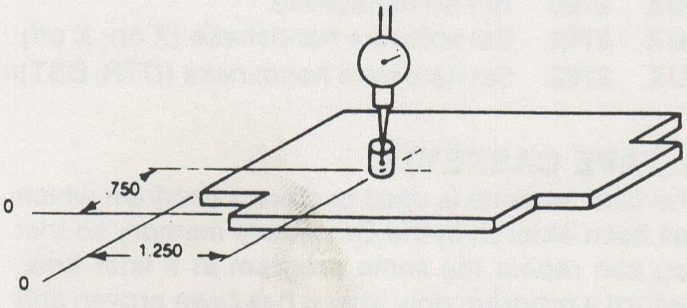


Figure 62: PRESETTING ZERO



In the top example, when the edge finder is at position A, the spindle center is -.100 from X absolute zero. While the spindle is at this position, press **MANUAL, X-.100, EVENT ENTER**. This will tell the control that the spindle center is not at zero, but at a position of X-.100 absolute and will set zero at the part edge. (.100 if for a .200 diameter edge finder. Use half the diameter of your edge finder if it is not .200) When at position B, press **MANUAL, Y-.100, EVENT ENTER**. This will set Y,O at the edge of the part. The minus (—) sign is used because this tells the control the absolute position of the spindle center.

In the bottom example, after the pin is indicated, zero can be preset in both axis. Press **MANUAL, X1.250, Y.750, EVENT ENTER** and zero will be set at the corner of the part. The dimensions are plus because this tells the control the absolute location of the spindle center.

E. MANUAL PROGRAMMED MOVES

There are times when you will want to move to a known position with the servo motors. You can use the manual JOG section as previously discussed or you can execute a "command in manual."

In figure 63, after the left hole is indicated, say you wish to rapid to the right hole and check position. With the motors on, press **MANUAL, X 3.250**, have the RAPID light in the programming keyboard lit, and press **START**. The machine will move to the commanded position as if you had executed it from a program. Alternatively, you could have pressed **X 1.750, INCR, START** and the machine would have moved to the same position.

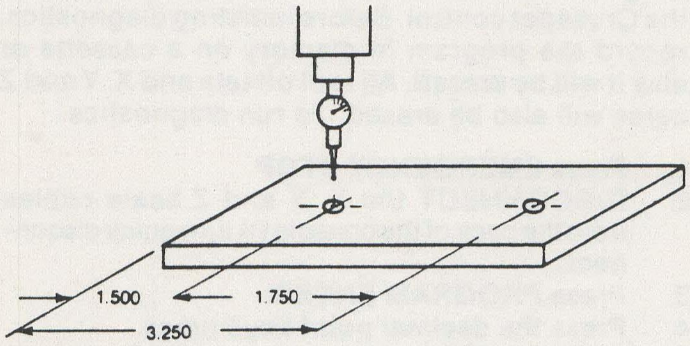


Figure 63: MANUAL PROGRAMMED MOVES

This feature is used quite often to recheck tool length offsets. First enter the offsets and execute or "read" the offset values in **SINGLE STEP** or **AUTO**. Then, to check the offset of tool 1, press **MANUAL, TOOL 1, START**. The offset value will appear in the Z axis display. By pressing **Z, O, ABS, START**, the tool will move to the point where the tool was referenced. Press **TOOL, O, START, Z O (ABS), START** to retract the quill. Each tool can be checked in this fashion.

F. PROGRAMMING FROM THE PART

After the outline of the pocket in figure 64 has been programmed, the center portion remains. Each clean-out pass could be programmed or the pocket could be finished using the handwheels, and at the same time the positions which are moved to can be entered in the program for the next part.

For example, say that the last move of the pocket programmed was Event 27. To enter the clean-out moves as they are executed with the handwheels:

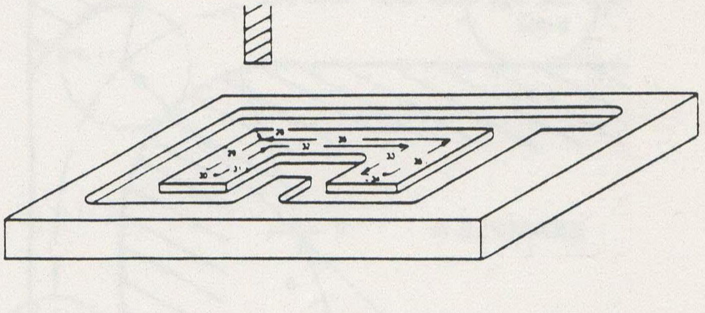


Figure 64: PROGRAMMING FROM THE PART

1. Press **MANUAL**, then press **CYCLE ENTER**. Both modes will be active.
2. Press **28, EVENT SEARCH** to make that event active.
3. Press **DISP POSN** (display position). This will show the absolute position of the X, Y and Z axis.
4. Move the machine to the next X Y position which you want to move to after Event 27.
5. Press **X, Y;** have the appropriate **RAPID/FEED** light on; have the **INCR** light off; and press **EVENT ENTER**. This will take the dimensions in the displays and enter them into the program.
6. Move the machine to the next desired position, press **X, Y, EVENT ENTER**. Repeat this procedure for each move.

G. FEEDRATE COMPENSATION

The size of the cutter and the size of the radius being cut will affect the feedrate which should be used. The programmer must change the feedrate in the program when constant surface speed or constant chip load is required.

In the example A in figure 65 a .375 cutter is used to cut outside a 1.000 part radius at 8 inches per minute. In order to maintain a constant surface speed at the edge of the cutter, the feedrate at the center of the cutter will have to be increased by using the formula:

$$\frac{\text{Part Radius} + \text{Cutter Radius}}{\text{part radius}} \times \text{Feedrate}$$

or

$$\frac{1.000 + .375}{1.000} \times 8 = 11 \text{ IPM}$$

The feedrate will have to be increased to 11 IPM at the part center to maintain 8 IPM at the edge of the cutter.

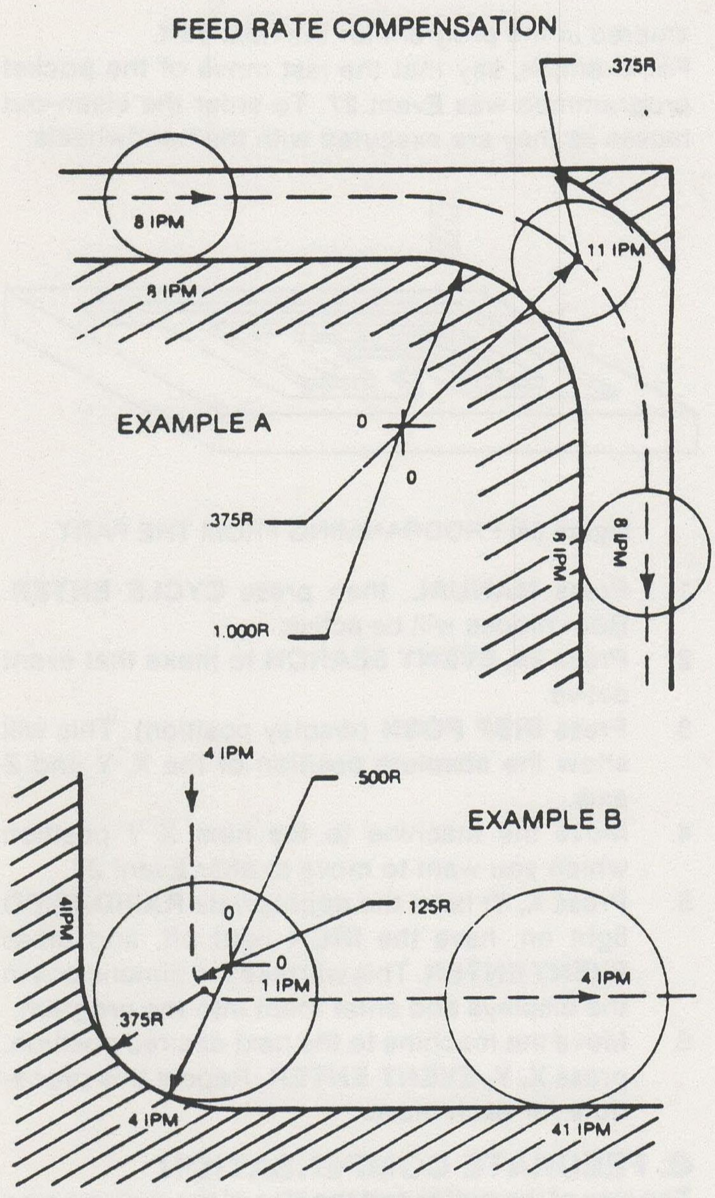


Figure 65:

The events in the program should be written in the following order:

1 **Feed 8.0**
2 **X O F A**
3 **Feed 11.0**
4 **Arc CW**
5 **X O Y O FA**
6 **X 1.375 Y O FA**
7 **Arc**
8 **Feed 8.0**

Notice that the feedrate is increased for the circular cut only. It is then changed back to 8 IPM for the straight line cut. The programmer is responsible for moving the cutter at the correct feedrate so the programmer must adjust the feed-rate in the program when necessary.

Example B in figure 65 shows the cutter path for an inside radius. A .375 R cutter is used for a .500 part radius at 4 inches per minute. (The drawing is 2 times scale for clarity.) To maintain a constant surface speed at the edge of the cutter, the feedrate at the center of the cutter must be decreased by using this formula:

$$\frac{\text{Part Radius - Cutter Radius}}{\text{part radius}} \times \text{feedrate}$$

or

$$\frac{.500 - .375}{.500} \times 4 = 1$$

The feedrate would have to be decreased to 1 IPM at the center of the cutter to maintain 4 IPM at the edge of the cutter.

Note that in the top example, the outside of the cutter is cutting material. In this case the feedrate should not be increased for a full cut (rough cut) but actually decreased because the outside is cutting. For a finish cut, when the outside edge is not being cut, the feedrate should be increased.

H. DIAGNOSTICS

Diagnostics (or self-test) is a standard feature of the Crusader control. Before initiating diagnostics, record the program in memory on a cassette **or else** it **will be erased. All** tool offsets and X, Y and Z zeros will also be erased. To run diagnostics.

1. **Press EMERGENCY STOP**
2. DISCONNECT the X, Y and Z scale cables from the back of the console (¼ turn quick disconnect).
3. Press **PROGRAM ENTER.**
4. Press the **decimal point key**5 times.
5. Press **START.**

The console should display changing numbers and lights on the keyboard as it goes through diagnostics. If the same number flashes continuously, a failure has been detected in the console. Contact your nearest Anilam distributor or call 1-800-327-6340. If no failure is detected, the diagnostic routine will end after approximately 1¼ minutes. A manufacturing code will be displayed in the readouts. To return to an operating condition after diagnostics, press **Decimal Point START.**

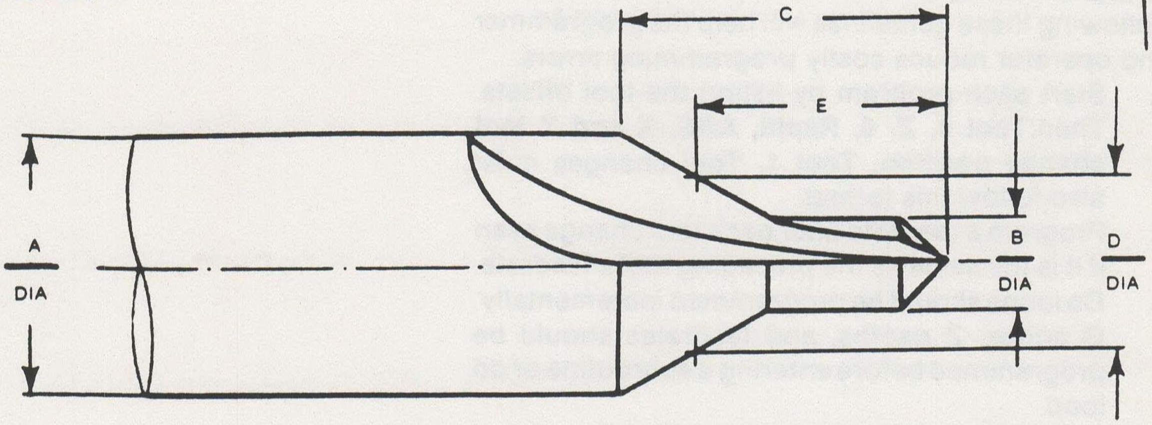
A keyboard check can be done immediately after diagnostics by pressing **Decimal Point EVENT ENTER.** The displays will go blank. Press the **X** button and "00" will be displayed. Press **7** and 01 will display, **8** will display 02, **9** will display 03, etc. Each button is numbered and can be tested to determine if it is functioning properly. To return to an operating condition, press **Decimal Point, START.**

I. GUIDELINES

Following these guidelines will help the programmer and operator reduce costly programming errors.

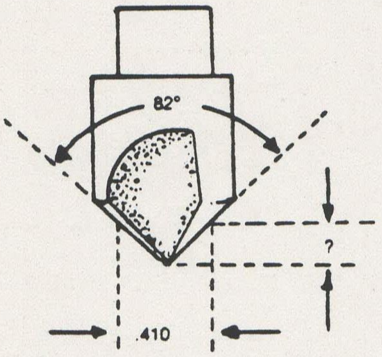
1. Start each program by listing the tool offsets. Then:**Tool 0, Z, 0, Rapid, ABS, X and Y tool change position, Tool 1.** Tool changes must also follow this format.
2. Program a feedrate after each tool change **even** if it is the same as the preceding tool's feedrate.
3. Do loops should be programmed incrementally. G codes, Z depths, and feedrates should be programmed before entering a subroutine or do loop.
4. Subroutines must be entered after the end of the program.
5. Start operation of the program only at the beginning or at the event number of the block containing Tool O before the tool at which you want to start.
6. After the final hole is drilled, cancel the canned cycle with G 80.
7. For circles which start and end on quadrants, incremental programming usually eliminates mathematical calculations.
8. Contours must have feedrates compensated to correct surface speed.
9. Program rapid X, Y and Z moves in separate blocks. When raising the spindle, move Z first then X, Y. When lowering the spindle, move X, Y then Z.
10. Write the program down and update the written copy when you edit.
11. Only arc centers, end points, incremental and absolute may be programmed between the arc CW (or CCW) and final arc blocks. Changes in feedrates or any other command must not be programmed between arc start and arc end commands.
12. Program tools in sequence always starting at number 1. Keep the written program and record the tool description on it. It is also helpful to record the rpm for each tool and the feedrate percentage for possible repeat orders. This information is also helpful for reference when programming other parts using similar tools.
13. Write incremental **or** absolute and rapid **or** feed for every program block that causes movement.
14. Think!

J. CENTER DRILL CHART

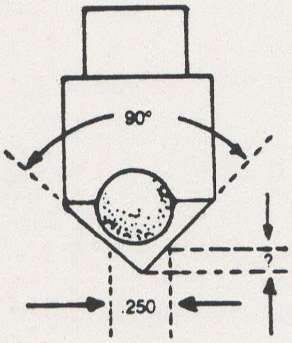


#	A	B	C	D	E	#	A	B	C	D	E	#	A	B	C	D	E
1	1/8	3/64	.128	1/16	.074	5	7/16	3/16 .187	.458	11/32	.377	7	5/8	1/4 .250	.650	1/2	.542
				3/32	.101					3/8	.404					17/32	.569
2	3/16	5/64 .078	.197	3/32	.116					13/32	.431					9/16	.586
				1/8	.143	6	1/2	7/32 .219	.525	1/4	.309					19/32	.623
				5/32	.170					9/32	.336	8	3/4	5/16 .312	.784	11/32	.433
3	1/4	7/64 .109	.265	1/8	.157					5/16	.363					3/8	.460
				5/32	.184					11/32	.390					13/32	.487
				3/16	.211					3/8	.417					7/16	.514
				7/32	.238					13/32	.444					15/32	.541
4	5/16	1/8 .125	.434	3/32	.289					7/16	.471					1/2	.568
				3/16	.316					15/32	.498					17/32	.595
				7/32	.343	7	5/8	1/4 .250	.650	9/32	.353					9/16	.622
				1/4	.375					5/16	.380					19/32	.649
				9/32	.397					11/32	.407					5/8	.676
5	7/16	3/16 .187	.458	7/32	.269					3/8	.434					21/32	.703
				1/4	.296					13/32	.461						
				9/32	.323					7/16	.488					23/32	.757
				5/16	.350					15/32	.515						

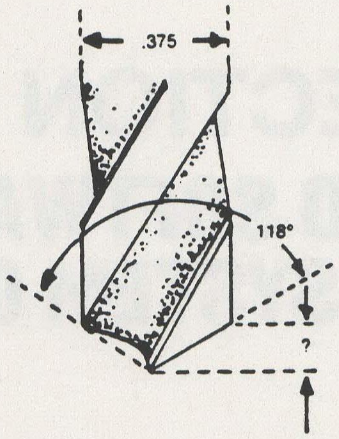
K. COMMON TOOL POINT FORMULAS



82° COUNTER SINK
TO CALCULATE THE DEPTH FOR A KNOWN DIAMETER:
MULTIPLY
DIAMETER X .575 = DEPTH



90° COUNTER SINK OR SPOT DRILL
TO CALCULATE THE DEPTH FOR A KNOWN DIAMETER:
MULTIPLY
DIAMETER X .5 = DEPTH



118° DRILL POINT
TO CALCULATE THE LENGTH OF A DRILL POINT, MULTIPLY
THE DRILL DIAMETER X .3 = POINT LENGTH

TO CHANGE INCHES PER REVOLUTION TO INCHES PER MINUTE, MULTIPLY
IPR X RPM = IPM

SECTION 11:

ADVANCED SOFTWARE SYSTEM ONE

ADVANCED SOFTWARE SYSTEM - I

The Advanced Software System - I package is an option that gives the operator canned cycles for bolt-hole patterns and rectangular and circular pocket milling. This package will also enable the operator to use features such as Polar Rotation and Polar Coordinates for easier positioning and reduced programming.

Each feature will be explained in detail in the following sections. The canned cycles will always use "variables" that allow the Crusader to make necessary calculations and complete the desired routines. These variables and their specific definitions are listed with the example for each ASS-1 feature.

Each ASS-1 feature is activated or deactivated through an assigned G-code number. A list of these codes and their function is below:

G10	Rapid polar move with spindle at polar center.
G11	Feed polar move with spindle at polar center.
G12	Rapid polar move with center specified.
G13	Feed polar move with center specified.
G51	Polar rotation (activate).
G52	Polar rotation (deactivate).
G53	Scaling (activate).
G54	Scaling (deactivate).
G75	Frame Milling
G76	Hole milling.
G77	Circular pocket milling.
G78	Rectangular pocket milling.
G79	Bolt hole circle drilling.

NOTE

Only one G-code can be active at a time with the exception of the G80 series codes. For example, Scaling cannot be used with Rotation, Polar Moves, Bolt Circles, etc., but could be used with the G80 series. All "V-codes" associated with these features must be input each time a G-code is used. Also, cutter compensation can be active with Rotation.

A. POLAR COORDINATES:

Polar Coordinates can be used when you need to move to a position that is dimensioned by an angle and a distance from the center of that angle (or, a radius).

See figure 66. To move from point A to point B using standard programming, the X and Y dimensions would have to be calculated. By using Polar Coordinates, the angle (30°) and radius (1.125) can be programmed and no calculations are needed. A polar coordinate is an alternative way of moving to a point when it is dimensioned as a radius and an angle.

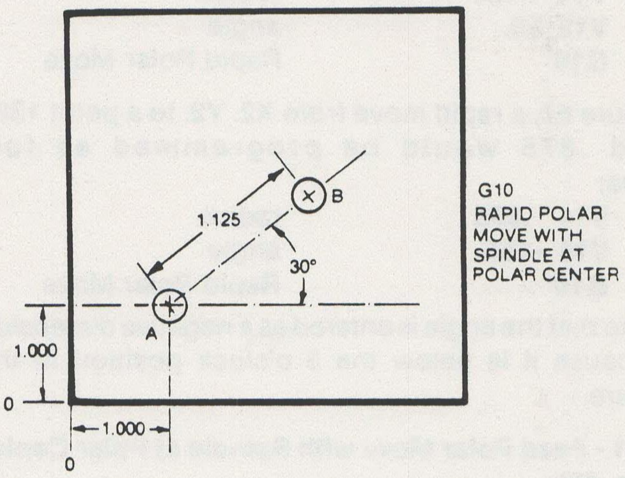


Figure 66

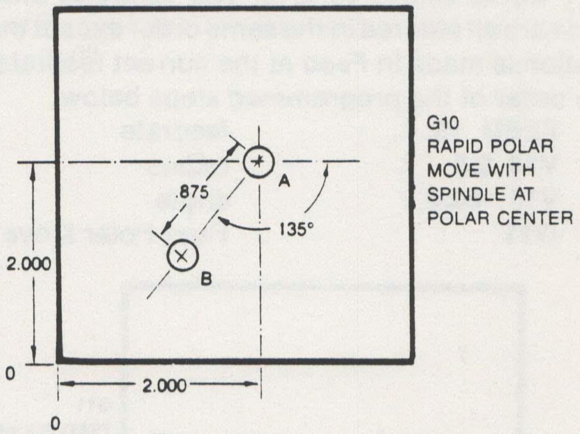


Figure 67:

With this understanding of Polar Coordinates, review each part example and explanation below. Each G-code and variable used to activate these features will be defined.

G10 - Rapid Polar Move with Spindle at Polar Center (Fig. 66 and 67):

You would use this G-code when the spindle is at the polar center (point A). You must enter the following two variables **before** programming the G-code. V14 (which is the radius dimension) and V15 (which is the angle from the 3 o'clock position).

NOTE

All angles must be measured from the 3 o'clock position in relationship to the polar center. A clockwise move from the 3 o'clock position is entered as a minus dimension. A counterclockwise move is a plus dimension.

In Figure 66, a rapid move from X1. Y1. to a point 30° and 1.125 would be programmed in the following order:

V14 1.125	radius
V15 30.	angle
G10	Rapid Polar Move

Figure 67, a rapid move from X2. Y2. to a point 135° and .875 would be programmed as follows:

V14 .875	radius
V15 -135.	angle
G10	Rapid Polar Move

Note that the angle is entered as a negative dimension because it is below the 3 o'clock position in the figure.

G11 - Feed Polar Move with Spindle at Polar Center (Fig. 68):

G11 works similar to G10. The variables and G11 code are all entered in the same order except the tool motion is made in Feed at the current feedrate. See the order of the programmed steps below:

FEED 15.	feedrate
V14 2.3	radius
V15 -63.5	angle
G11	Feed Polar Move

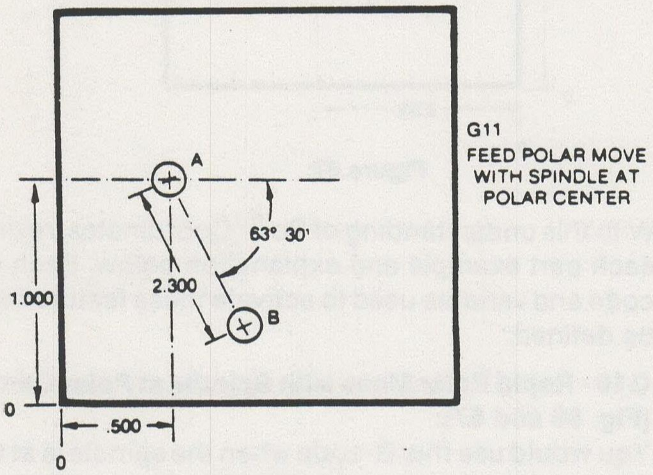


Figure 68:

G12 - Rapid Polar Move with Center Specified (Fig. 69):

This G-code works when you need to make a polar move but the present tool location is not at the polar center. Using the assigned V-codes you must program the location of the polar center. These V-codes must be entered prior to programming G-12. Review the program steps below. This will show how to move from X3. Y1. to the point at 135° from the polar center (point A to B).

V11 1.	Polar center X
V12 1.5	Polar center Y
V13 0	Angle measured from 0°
V14 .6	Radius
V15 135.	Angle
G12	Rapid move

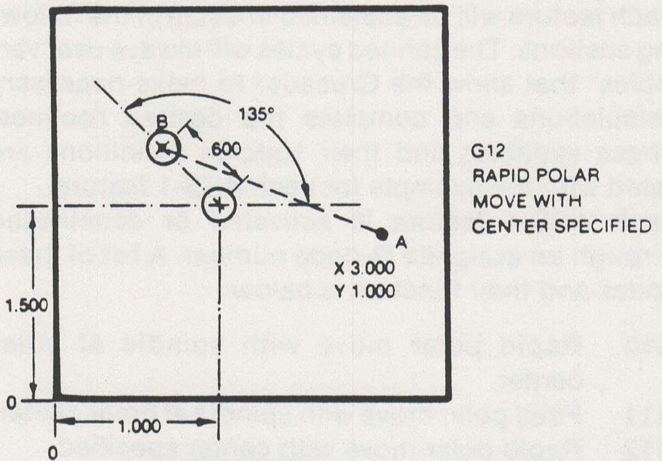


Figure 69

G13 - Feed Polar Move with Center Specified (Fig. 70):

G13 works similar to G12, except the move is made in FEED at the current feedrate. The move from X1. Y0 to the point 15° 15' from the Index Angle of the 90° to polar center (point A to B) is programmed as follows:

FEED 20.	feedrate
V11 2	center X
V12 1.	center Y
V13 90.	angle measured from 0°
V14 .4375	radius
V15 -15.25	angle
G13	feed move

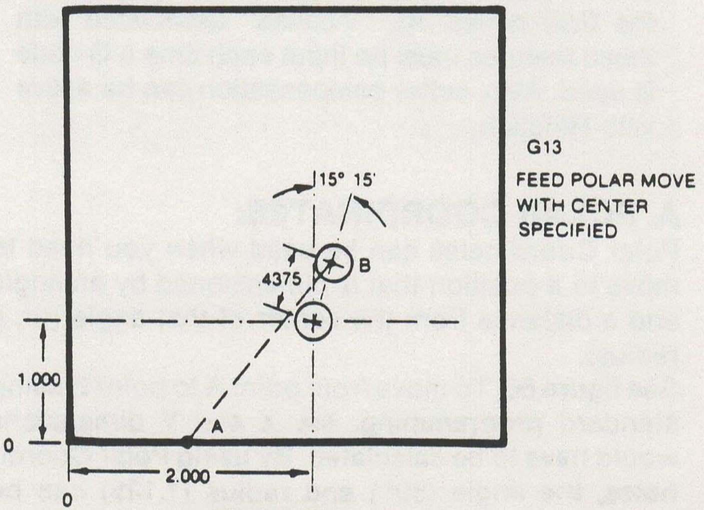


Figure 70

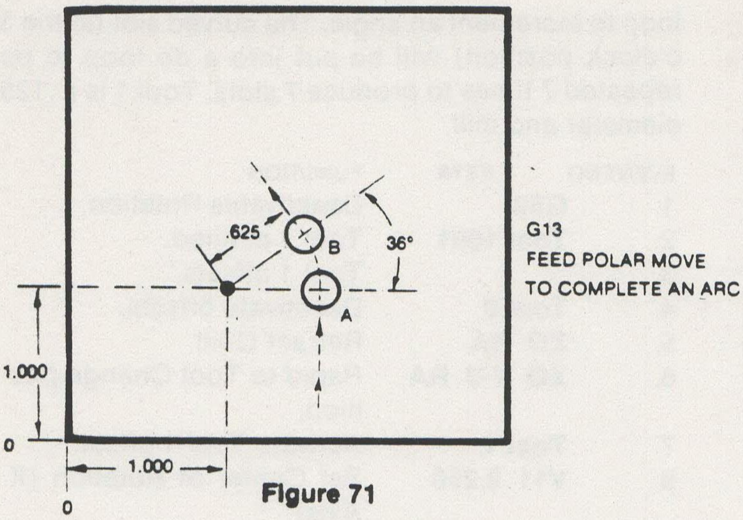


Figure 71

You may also use G13 so that the end point of an arc can be reached with a polar move. The format for making the arc is the same as it would be if you knew the XY coordinates of the end point. However, instead of programming the XY end point, the polar command G13 is used. See figure 71 and refer to the program steps below to see this capability in use.

FEED 5.	Feedrate
X1.625 Y1. FA	Feed to start of arc
V11 1.	Center X
V12 1.	Center Y
V13 0.	Angle measured from 0°
V14 .625	Radius
V15 36.	Angle
ARC CCW	Arc Command CCW
X1. Y1. FA	Arc Center
G13	Polar move to end point
ARC	Arc Finish Command

Because the Arc CCW Command is active, the tool will move in a circular rather than linear motion.

B. POLAR ROTATION:

Polar Rotation is used when you need to rotate a program, or just a portion of a program, around a point. The operator can rotate holes, slots or milled shapes around a point.

For example, in figure 72, it is necessary to mill 3 pockets as shown. Once the dimensions for pocket A are programmed, the operator can use Polar Rotation twice to rotate the pocket 120° around the center point. This would eliminate the need to calculate and program the dimensions of pockets B and C.

G51 and G 52 - Polar Rotation (Activate and Deactivate):

G51 causes every motion command to be rotated until a G52 is reached in the program. NOTE: The first XY move after G51 must be an absolute move. The following variables must be entered before programming a G51: V11 (Polar Center X), V12 (Polar Center Y),. V13 (Index Angle of Rotation), and V15 (Angle Rotated). V11 and V12 will describe the point which the program will rotate around. V13 describes the angle from which the program will rotate.

In figure 72, the operator would put the dimensions of slot "A" in a subroutine (Sub 1). The program to execute and rotate the slot to positions "B" and "C" would be as follows:

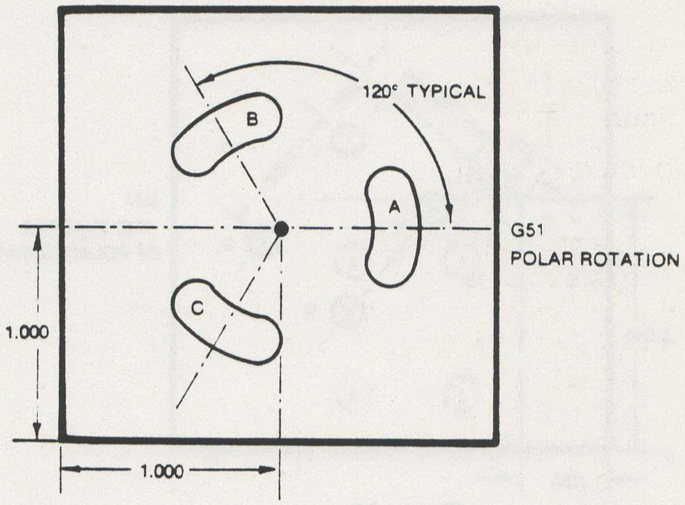


Figure 72

CALL 1		Execute Sub 1-Position A
V11	**1.**	Center X
V12	**1.**	Center Y
V13	**0.**	Index Angle
V15	**120.**	Angle of Rotation: 120°
G51		Activate Rotation
CALL 1		Execute Sub 1-Position B
V11	**1.**	Center X
V12	**1.**	Center Y
V13	**0.**	Index Angle
V15	**-120.**	Angle of Rotation: -120°
G51		Activate Rotation
CALL 1		Execute Sub 1-Position C
G52		Deactivate Rotation

See figure 73. The drilled holes in this example are to be rotated 45° from positions 1, 2, 3, 4 to positions A, B, C, D. The program below will explain the procedure:

V20	**10.**	Feedrate for drilling
V21	**.1**	Drilling Start Height
G81		Activate Drilling Canned Cycle

V11	**1.**	Center of Rotation - X
V12	**2.**	Center of Rotation - Y
V13	**0.**	Index Angle
V15	**45.**	Angle of Rotation
G51		Activate Rotation
X1.125 Y1.5 Z-.3 RA		XY Coordinate and Z-depth of hole 1
Y-1. RI		Location of Hole 2
X.750 RI		Location of Hole 3
Y1. RI		Location of Hole 4
G52		Deactivate Rotation
G80		Deactivate G81

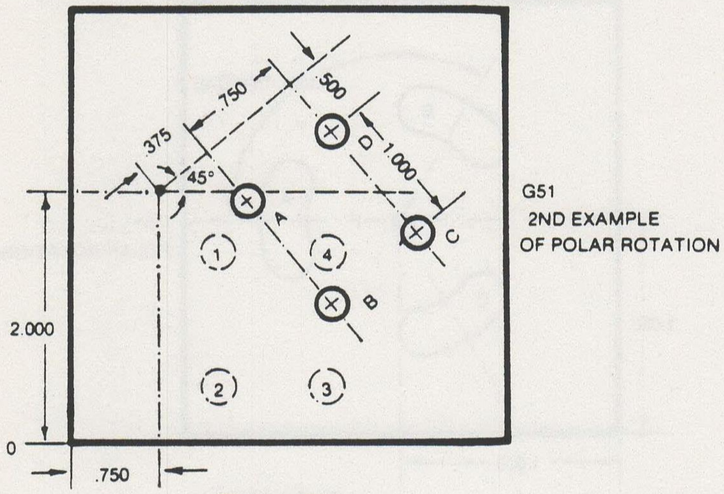


Figure 73

Note in this example that the 4 holes were to be rotated only once. It is not necessary to create a subroutine. If the 4 holes were to be rotated more than once, it would be wise to put the 4 hole dimensions in a subroutine. That would then reduce the programming steps.

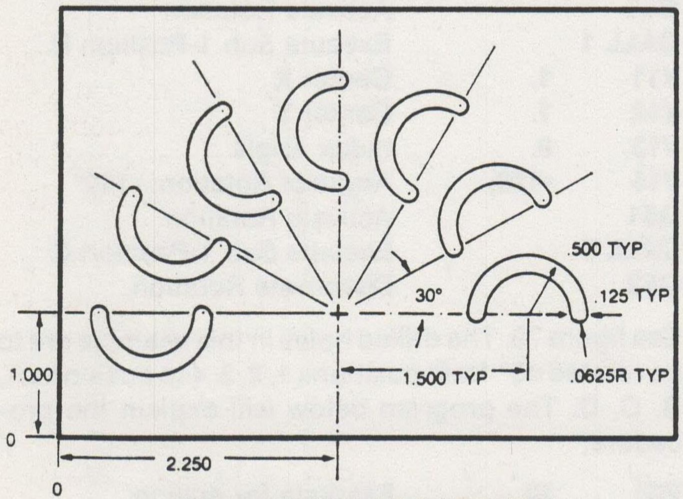


Figure 74: POLAR ROTATION

The program section written below for figure 74 shows how Polar Rotation can be used with a do loop to increment an angle. The curved slot (at the 3 o'clock position) will be put into a do loop to be repeated 7 times to produce 7 slots. Tool 1 is a .125 diameter end mill.

EVENT NO	KEYS	FUNCTION
1.	**G52**	Deactivates Rotation.
2.	**Tool 1001**	Tool 1 defined.
3.		Tool 1 offsets.
4.	**Tool 0**	Deactivate offsets.
5.	**ZO RA**	Retract Quill.
6.	**XO Y-3 RA**	Rapid to Tool Change position.
7.	**Tool 1**	Activate Tool 1 offset.
8.	**V11 2.250**	Set Center of Rotation (X Axis).
9.	**V12 1.000**	Set Center of Rotation (Y Axis).
10.	**V13 O**	Set Index Angle.
11.	**V15 0**	Set Angle Rotated (See Note).
12.	**DO 7**	Activate do loop repeated 7 times.
13.	**G51**	Activate Rotation.
14.	**X3.3125 Y1. RA**	Rapid to end of slot.
15.	**Z.1 RA**	Rapid Z to .1 above part.
16.	**Feed 10.0**	Set Feedrate.
17.	**Z-.05 FA**	Feed Z into part.
18.	**ARC CW**	Start ARC Clockwise.
19.	**X.4375 YO FI**	Define ARC Center.
20.	**X.875 YO FI**	Define ARC end point.
21.	**ARC**	ARC end statement.
22.	**Z.1 RA**	Rapid Tool .1 above part.
23.	**V13 30.**	**Adjust** V13 by 30°.
24.	**END**	End of do loop.
25.	**G52**	Deactivates Rotation.
26.	**Tool O**	Deactivates offsets.
27.	**ZO RA**	Retract Quill.
28.	**XO Y-3 RA**	Rapid to Tool Change Position.
29.	**END**	End of Program.

NOTE:

The do loop started with V13 and V15 set to zero because the first slot is not rotated. Event 13 activates Rotation but the slot is rotated° in the first loop.

Event 23 adjusts V13 to 30 degrees. Starting with the second loop, each slot is then rotated 30 degrees because G51 is activated.

Rotation can take place in the X and Y axis only. A Z axis move cannot be rotated.

C. SCALING

This feature of the Advanced Software System - I package will allow the operator to scale the entire program, or just a portion of a program, up or down to affect the size of the part being made.

Part scaling is activated by the code G53. However, the following variables must be entered before G53 is programmed: V11 (Scale Center X), V12 (Scale Center Y), V13 (Scale Center Z), V16 (Scale Factor X), V17 (Scale Factor Y), and V18 (Scale Factor Z). Bear in mind that the first XY move after programming the G53 code must be an absolute move.

Figure 75 shows the dimensions of a large box to be scaled down to a smaller box. The steps below show how this should be programmed into the Crusader.

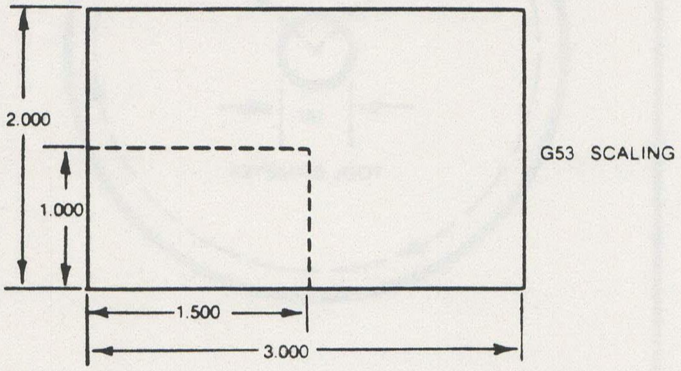


Figure 75

V11	**O**	Center X.
V12	**O**	Center Y.
V13	**O**	Center Z.
V16	**.500**	Scale Factor X.
V17	**.500**	Scale Factor Y.
V18	**1.**	Scale Factor Z.
G53		Activate Scaling.
XO YO FA		
X3. FA		Original Coordinates to be scaled
Y2. FA		
XO. FA		
YO FA		
G54		Deactivate Scaling.

Figure 76, and the program steps below, show how the operator may scale each individual axis with different scale factors. This example uses dimensions scaled from a certain point.

> **NOTE:**
> **Scaling of circles is permitted if the scale factor for each axis is the same.**

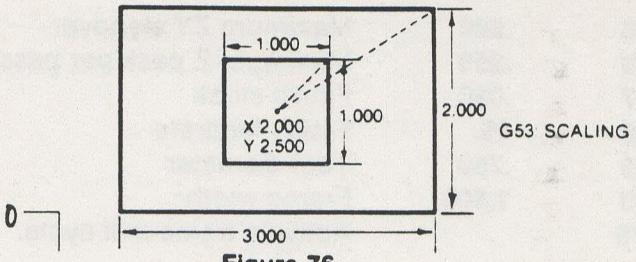


Figure 76

Note: Set zero is actually X.5, YI.5 from bottom left corner of part.

Feed	**20.**	Activate Feedrate.
X2. Y2.5	**RA**	Move tool to center.
V11	**2.**	Central X.
V12	**2.500**	Center Y.
V13	**O**	Center Z.
V16	**3.**	Scale Factor X.
V17	**2.**	Scale Factor Y.
V18	**1.**	Scale Factor Z.
G53		Activate Scaling.
X2.5 Y3.	**FA**	Original Coordinates to be Scaled.
Insert Z depth here if necessary.		
Y-1. FI		
X-1. FI		
Y1. FI		
X1. FI		
G54		Deactivate Scaling.

D^1. FRAME MILLING CYCLE

The programmer uses this canned cycle to mill a rectangular frame. The following variables must be entered before G75 (activate frame milling cycle) is programmed. V40 (Z axis dimension .100 above the top surface of the part to be milled into), V41 (the incremental X distance of the inside of the frame), V42 (the incremental Y distance of the inside of the frame), V43 (absolute Z depth of the frame), V44 (corner radius on the inside of the frame), V45 (maximum incremental step over for each pass in X and Y), V46 (maximum incremental Z peck depth per pass), V47 (stock to be left on sides and bottom for finish pass), V48 (feedrate for finish pass), V49 (tool diameter), V50 incremental width of frame.

The operator must first position the center of the tool over the lower left inside corner of the frame. The program shown below will machine the frame shown in Figure 76A.

X2.000	**Y2.000 RA**	Rapid to start point 2
Feed	**10.**	Roughing feedrate
V40	**.100**	Starting height
V41	**3.000**	Length X
V42	**2.500**	Width Y
V43	**-.500**	Depth Z
V44	**.500**	Corner radius

V45	**.500**	Maximum ZY stepover
V46	**.250**	Maximum Z peck per pass
V47	**.020**	Finish stock
V48	**15.**	Finish feedrate
V49	**.750**	Tool diameter
V50	**1.500**	Frame width
G75		Activate frame mill cycle.

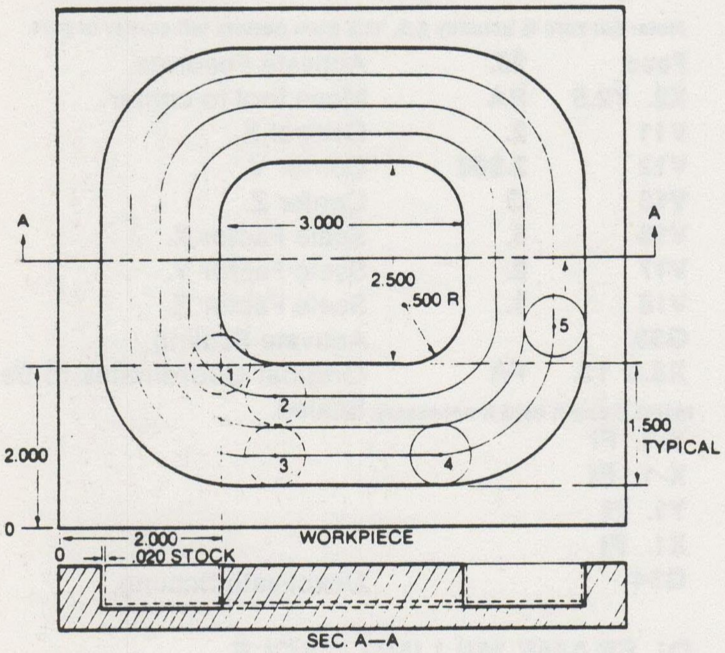


Figure 76A

The operator programs the tool to move to position 1. Upon execution of G75 the tool will rapid to the V40 dimension, rapid to position 2, slope to position 3 at the first Z peck depth, feed to position 4, 5 and continue around the frame back to position 3, feed towards position 2 for the next pass around the part until the full width of the frame (minus the finish stock) is machined. The pattern is repeated until the finish depth (minus the finish stock) is reached. The finish pass is then made at the V48 finish feedrate.

D. HOLE MILLING CYCLE:

The Hole Milling Cycle code (G76) can be used to mill a hole **through** a part or to do a counter bore. The operator must enter the following variables before G76 is programmed: V18 (hole diameter) and V49 (tool diameter).

The operator must position the tool at the center of the hole and have it placed at the required depth. When G76 is executed, the tool will follow the path shown in figure 77. The tool will first move 45°, then approach the hole diameter on a tangent, contour 360°, leave on a tangent, and then move back to the center on a 45° angle.

To mill the hole shown in figure 77, put the tool in the center of the hole location and at the correct depth, then follow the program steps below:

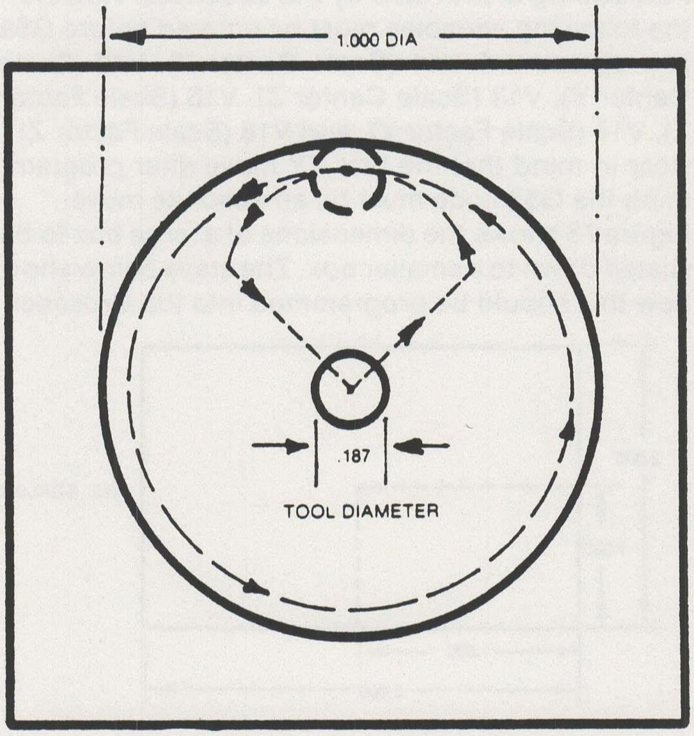


Figure 77: HOLE MILLING CYCLE (G76)

V18	**1.000**	Hole diameter
V49	**.187**	Tool diameter
G76		Activate Hole Diameter

E. CIRCULAR POCKET MILLING:

This cycle, which is activated by a G77 code, is used to mill circular pockets and eliminate extensive programming on the part of the operator.

The operator must enter the following variables before G77 is programmed: V40 (Z axis dimension .100 above the top surface of the pocket area), V41 (Pocket Center X), V42 (Pocket Center Y), V43 (Pocket Depth Z), V44 (Pocket Diameter), V45 (Maximum incremental step-over for each pass), V46 (Maximum incremental depth of each pass), V47 (Stock to be left over for finish cut on sides and bottom), V48 (Feed rate for finish cut), V49 (Tool Diameter).

The program to cut the pocket shown in figure 78 would be written as follows:

Feed	**10.**	Roughing Feed Rate.
V40	**.100**	Starting Height.
V41	**2.250**	Pocket Center X. ABS

V42	**1.750**	Pocket Center Y. ABS
V43	**-.5**	Pocket Depth Z. ABS
V44	**2.**	Diameter of Pocket.
V45	**.2**	Maximum XY Step-over.
V46	**.25**	Maximum Z depth per pass.
V47	**.020**	Stock left for finish cut.
V48	**15.**	Feed rate for finish cut.
V49	**.250**	Tool Diameter
G77		Activate Circular Pocket Milling

Upon execution, the tool will rapid to the X Y center (Point 1) and rapid to the V40 dimension. It will then feed to the top of the work piece, slope to the edge of the pocket (Point 2) - minus finish stock - at the first depth. The tool will then contour 360 °, step over to point 3, contour 360° until the center is reached. If more roughing cuts are required, the slope-in and peck-over sequence is repeated. After the pocket is roughed out, the finish feed rate becomes active and the finish cut is taken on the side and bottom. The tool will stop at the center of the pocket and return to the V40 dimension. The V40 dimension must be .100 (2mm) above the top of the workpiece. To alter this distance see the note concerning V99 at the end of section 7. This method of altering this distance can be used for all pocket milling cycles.

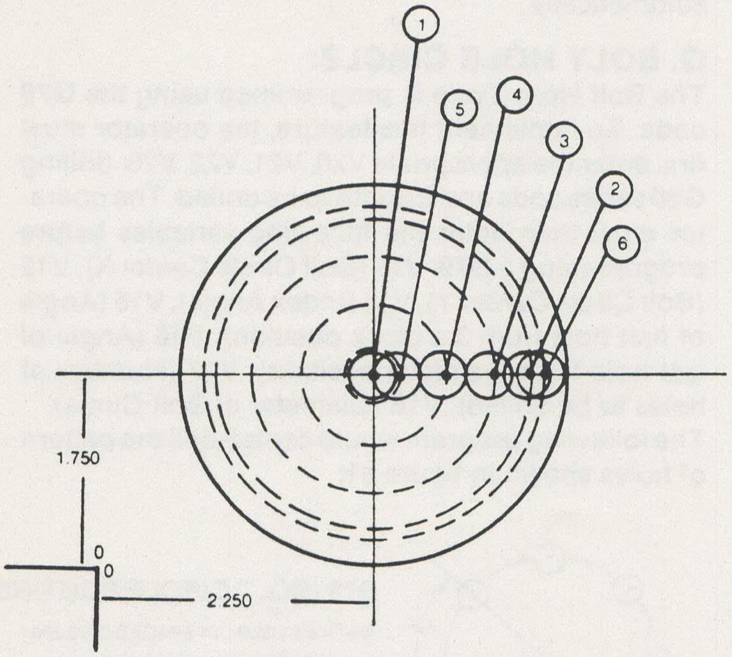
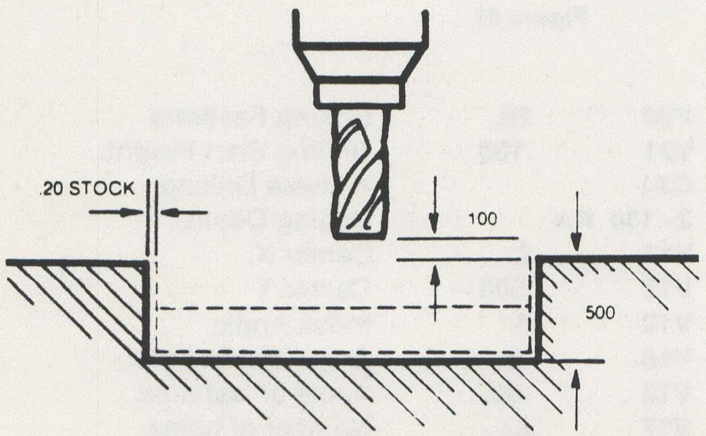


Figure 78: (G77)

POCKET 2 000 DIA (X 5 DEEP)
TOOL .250 DIA



- - - - - -

NOTE:

The tool will pause for a few seconds upon execution of G77 - G79 because the Control is doing calculations. If an unprogrammed dwell occurs, then an error has been made in the input data format.

F. RECTANGULAR POCKET MILLING.

The operator uses this cycle (activated by G78) to mill rectangular pockets without extensive programming. The operator must enter the following variables before programming the G78 code: V40 (Z axis dimension .100 above the top surface of the pocket), V41 (X axis incremental length of pocket), V42 (Y axis incremental length of pocket), V43 (Z axis absolute depth of pocket), V44 (pocket corner radius), V45 (incremental maximum step over for each pass in X and Y), V46 (incremental maximum Z peck depth for each pass), V47 (stock to be left on sides and bottom for finish pass), V48 (finish pass feed rate, V49 (tool diameter), V50 (set = to zero if a finish pass is required, set = to 1. if no finish pass is required). The operator must first position the center of the tool at the lower left corner of the pocket. The program shown below will machine the pocket shown in figure 79:

Feed	**8.0**	Roughing Feed Rate
V40	**.100**	Starting Height .100 above part.
V41	**4.500**	Length in X axis (incremental)
V42	**3.375**	Width in Y axis (incremental)
V43	**-.500**	Depth in Z axis (absolute).
V44	**.375**	Corner Radius.
V45	**.350**	Maximum X and Y Step Over
V46	**.250**	Maximum Z peck per pass.
V47	**.020**	Finish Pass Stock.
V48	**15.**	Finish Feed Rate
V49	**.500**	Tool Diameter

V50	**0**	Complete finish pass.
G78		Activate Pocket Milling.

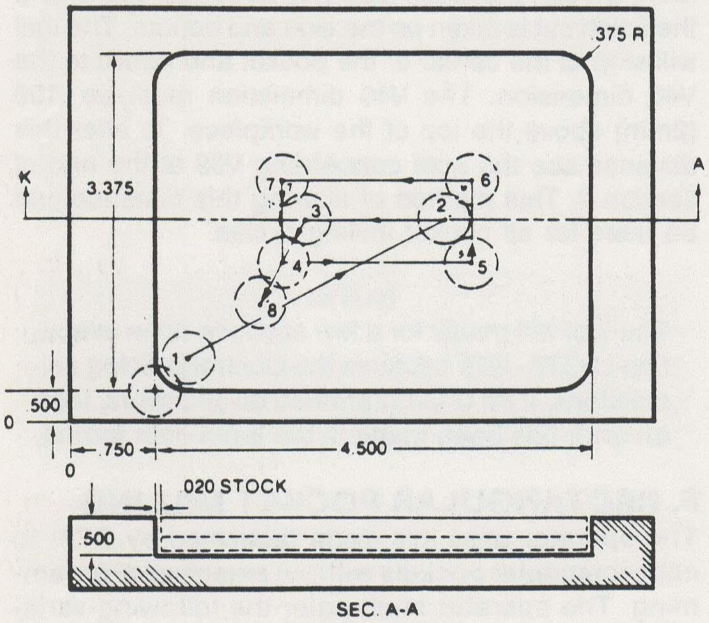


Figure 79

NOTE:

V45 (which is the step-over) should not be greater that 70% of the tool diameter or the tool will not clear all of the pocket. See Figure 80.

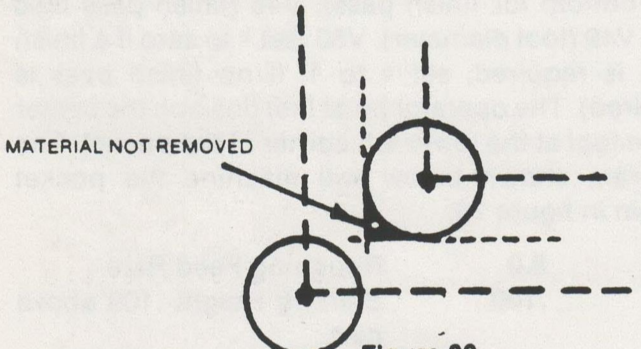


Figure 80

Upon execution of G78, the tool will rapid to the center of the corner radius (position 1). It will then rapid to the V40 dimension, feed to the top of the work piece, slope in 3 axis to position 2 to the first peck depth. The tool then machines the pocket by moving to 3, 4, 5, 6, 7, 4, 8, 9, 10, 11 etc., until the tool is at the pocket edge (minus finish pass stock).

This sequence will be repeated until the final depth (minus the finish pass) is reached. The finish feed rate will then become active and the final finish pass taken. The tool will retract to the V40 dimension and move to the lower left corner that it started from.

If V45 is entered as a plus value, the tool will move in a climb mill motion from 2, 3, 4, 5, 6, 7 etc. If V45 is entered as a negative value, the tool will move in a direct or conventional mill motion from 2, 3, 4, 7, 6, 5, 4 etc. The finish pass will always be a climb mill cut. The control will always adjust the maximum step over to make each pass equal. The Z peck will also be adjusted to make each peck equal. For instance, in this example, the maximum peck distance is .250, the total depth .500 and the finish .020. The control will make two Z pecks at .240 each, then the finish .020 pass.

If a corner radius of .010 or less is calculated, a straight line move at 45 ° will be made instead of a contour; otherwise the corner radii will be contoured automatically.

G. BOLT HOLE CIRCLE:

The Bolt Hole Cycle is programmed using the G79 code. To implement this feature, the operator must first enter the appropriate V20, V21, V22, V23, drilling G80 series code and Z depth to be drilled. The operator must then enter the following variables **before** programming a G79: V11 (Bolt Circle Center X), V12 (Bolt Circle Center Y), V13 (Index Angle), V15 (Angle of first hole from 3 o'clock position), V16 (Angle of last hole from 3 o'clock position), V17 (Number of holes to be drilled), V18 (Diameter of Bolt Circle).

The following program would center-drill the pattern of holes shown in figure 81:

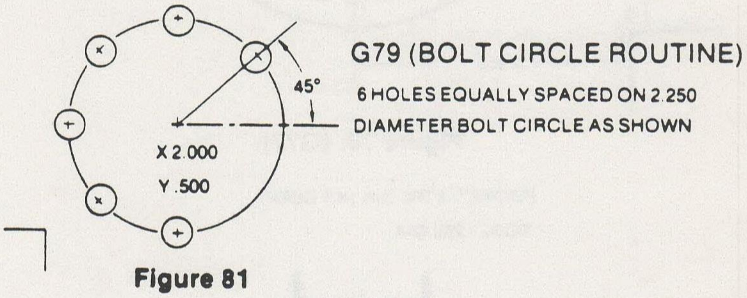

G79 (BOLT CIRCLE ROUTINE)
6 HOLES EQUALLY SPACED ON 2.250
DIAMETER BOLT CIRCLE AS SHOWN
45°
X 2.000
Y .500

Figure 81

V20	**10.**	Drilling Feedrate
V21	**.100**	Drilling Start Height.
G81		Activate Drilling.
Z-.130 RA		Drilling Depth.
V11	**2.**	Center X.
V12	**.500**	Center Y
V13	**0.**	Index Angle.
V15	**45.**	Angle of first hole.
V16	**-90.**	Angle of last hole.
V17	**6.**	Number of holes.

V18	**2.250**	Diameter of Bolt Circle.
G79		Activate Bolt Circle Routine
G80		Cancel drilling.

NOTE:
The tool will always move in a counter clockwise direction around the bolt circle, and the holes will always be equally spaced between the start angle and finish angle. The tool can be at any position when G79 is activated. Since G81 is active, the tool will not drill a hole at its present location when the Z depth is given, but will drill a hole only when the next X and/or Y location is reached.

H. ABSOLUTE ZERO SHIFT (AUX 1101)

This feature allows the programmer to move absolute zero from point to point on the table. When the operator is using several vises or is holding several workpieces in a large fixture then the Absolute Zero Shift feature is useful. The operator only has to program the part once then, after each program is complete, simply shifts the absolute zero point to the next workpiece and runs the program again.

In figure 82, the original absolute zero is set at the lower left corner of the lower left part. The absolute or incremental co-ordinates of the hole locations on that part are in subroutine 1. The program to drill the first part and then move the zero to the new locations is:

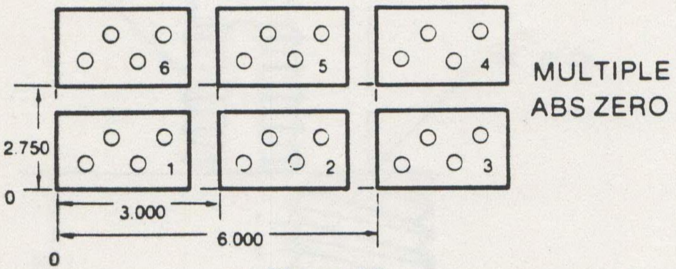


Figure 82

V20 10.	Sets drilling feed rate
V21 .1	Sets drilling start height
G81	Activate drilling
Z -.5	Drilling depth
Call 1	Drill Part 1
AUX 1101	Reset Absolute Zero
X3. Y0 Z-1.123	Set zero at Part 2 (Z = TLO)
Call 1	Drill Part 2
AUX 1101	Reset Absolute Zero
X6. Y0 Z-1.123	Set zero at Part 3 (Z = TLO)
Call 1	Drill Part 3
AUX 1101	Reset Absolute Zero
X6 Y2.750 Z-1.123	Set zero at Part 4 (Z = TLO)
Call 1	Drill Part 4
AUX 1101	Reset Zero
X3. Y2.750 Z-1.123	Set zero at Part 5 (Z = TLO)
Call 1	Drill Part 5
AUX 1101	Reset Zero
X0. Y2.750 Z-1.123	Set zero at Part 6 (Z = TLO)
Call 1	Drill Part 6
AUX 1101	Reset Zero
X0 Y0 Z-1.123	Reset zero at Part 1 (Z = TLO)
G80	Deactivate drilling

Changing tool numbers will reset absolute zero back to the tool 0 original position. Each time a tool is changed, AUX 1101 and the new zero location must be reset.

I. SOFTWARE LIMITS

Software Limits allow the programmer to define a "box" from which the tool cannot move out. Collisions between vises, fixtures and tooling, along with damage to the machine itself, can be avoided. Once these limits are set and activated, the tool will not move past them in any mode. Software Limits can be used as a safety stop to avoid drilling into the machine table, fixture or parallels. If a part overhangs the table, limits can be set to stop it from crashing into the column of the machine. If the operator tries to move past the limits set, the drive motors will cease to move the machine.

The program to set software limits for the part in figure 83 is:

V01 .600	(X low limit)
V02 14.400	(X high limit)
V03 .600	(Y low limit)
V04 7.400	(Y high limit)
V05 -1.300	(Z low limit)
V06 10.000	(Z high limit)
AUX 1111	(Activate software limits)

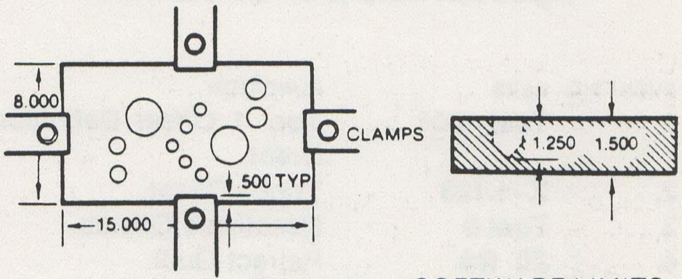


Figure 83 SOFTWARE LIMITS

After the job which uses these software limits is completed, they can be deactivated by programming AUX 1110. When AUX 1111 is programmed, the spindle **must be inside** the limits being set. If a software limit is reached, the program will stop. The operator must go into the manual mode and jog away from the software limit or use the handwheels to move away from the software limit. Depending on how fast you are moving when the software limit is reached, the tool may move up to .015 past the software limits.

J. HELICAL INTERPOLATION:

Helical Interpolation is used to move the X and Y axis in a circular motion and the Z axis up or down in a straight line, all at the same time. Thread milling or fan-blade milling are among the most common uses for helical interpolation.

The program written below will move the tool from point A to point B in helical motion as shown in figure 84.

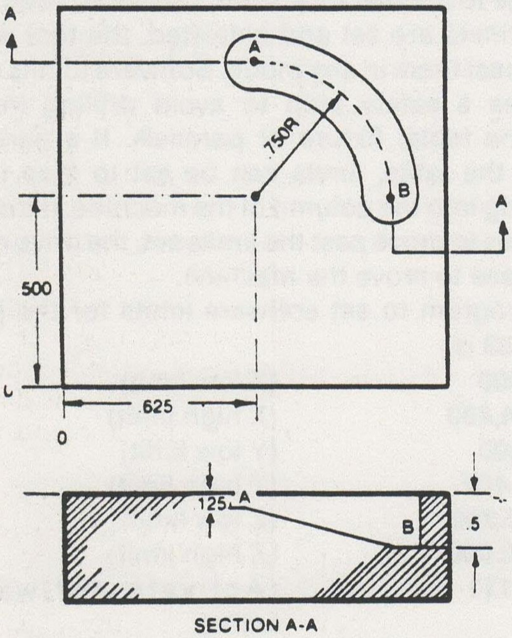


Figure 84: HELICAL INTERPOLATION

EVENT NO.	**KEYS**	FUNCTION
1.	**Tool 1001**	Tool 1 Offset Definition Event.
2.	**Z -1.123**	Tool 1 Offset.
3.	**Tool 0**	Deactivate Offsets.
4.	**Z0 RA**	Retract Quill.
5.	**X0 Y0 RA**	Rapid to Tool Change position.
6.	**Tool 1**	Activate Tool 1's Offset.
7.	**X.625 Y1.250 RA**	Rapid to Point A
8.	**Z.05 RA**	Rapid Tool .05 above part.
9.	**Feed 5**	Sets Feedrate.
10.	**Z-.125 FA**	Feed Tool to start depth.
11.	**ARC CW**	Described ARC Direction
12.	**X.625 Y.5 FA**	Describes ARC center.
13.	**V42 0.**	Sets the number of complete 360° revolutions to be done. Since only 90° is being cut in this example, V42 is set to zero.
14.	**X1.375 Y.5 Z-.5 FA**	Describes ARC end point in all 3 axis.
15.	**ARC**	ARC finish command
16.	**Tool 0**	Deactivates offsets.
17.	**Z0 RA**	Retracts Quill.
18.	**X0 Y0**	Returns to part change position.
19.	**END**	Ends program.

In the next example (fig. 85) the tool will move in helical motion to produce a thread with a thread milling cutter. Five threads per inch will be produced by starting and ending at the same X Y coordinate, setting the number of revolutions at 5 and moving the Z axis one inch.

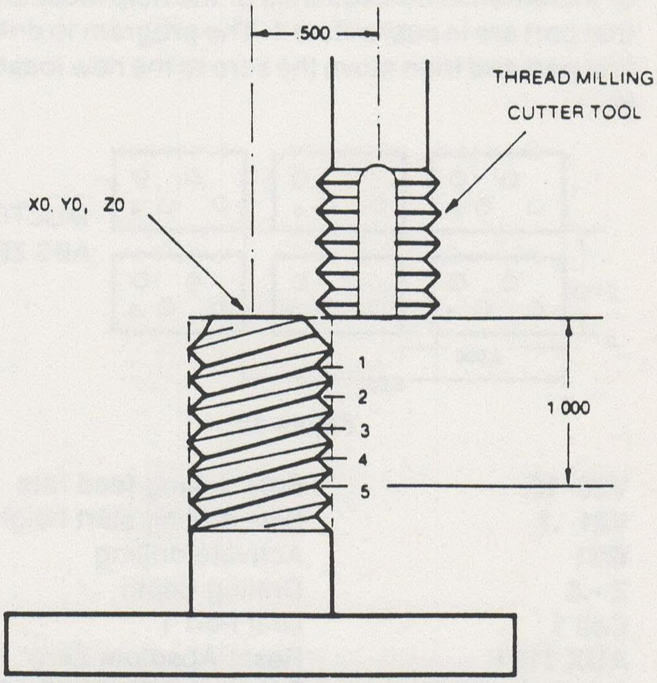


Figure 85: HELICAL INTERPOLATION

EVENT NO.	**KEYS**	FUNCTION
1.	**Tool 1001**	Tool 1 Offset Definition Event.
2.	**Z-1.123**	Tool Offset.

3.	**Tool 0**	Deactivate Offsets.
4.	**Z0, RA**	Retracts Quill.
5.	**X-2. Y 2.**	Rapid to tool change position.
6.	**Tool 1**	Activate Tool 1's Offset.
7.	**X.5 Y0 RA**	Rapids to XY Start Position.
8.	**Z.05 RA**	Rapids to .05 above part.
9.	**Feed 3.0**	Sets Feed Rate.
10.	**Z0 FA**	Feed Z to top of part.
11.	**ARC CW**	Describes ARC Direction.
12.	**X0 Y0 FA**	Describes ARC Center.
13.	**V42 5.**	Sets Complete 360° Revolutions to 5.
14.	**X.5 Y0 Z-1. FA**	Describes ARC end point in 3 axis.
15.	**ARC**	ARC finish command
16.	**Y-.5 FA**	Feeds tool off part.
17.	**Tool 0**	Deactivates tool offsets.
18.	**Z0 RA**	Retracts Quill.
19.	**X-2. Y2. RA**	Rapids to tool change position.
20.	**END**	Ends Program.

K. V CODE FORMAT RULES:

V01	Sets X axis low software limit. V01 thru V06 are absolute values from the original absolute zero. These limits do not shift when a multiple absolute zero is used.
V02	Sets X axis high software limit.
V03	Sets Y axis low software limit.
V04	Set Y axis high software limit.
V05	Set Z axis low software limit.
V06	Set Z axis high software limit.
V11	Polar center in the X axis for polar moves, bolt circle, or scaling. This value must always be absolute.
V12	Polar center in the Y axis for polar moves, bolt circle, or scaling. This value must always be absolute.
V13	Index angle used with polar rotation. When used with rotation, this value must always be an incremental angle. When used with G10-G13 polar moves, it is the angle which the polar angle is measured from. A positive value indicates a counter clockwise rotation. A negative number indicates a clockwise rotation. The range of values is + .01 to +360.00. Degrees are measured in decimal degrees, not minutes and seconds. An angle of 15° 30' must be entered as 15.50 degrees. This applies to all variables which describe angles. When Scaling is being used, V13 is center of the scaling in the Z axis and is always absolute.
V14	Radius for polar moves. This value must always be a positive.
V15	Angle for polar moves or angle of the first hole in a bolt circle. This angle is always measured from the 3 o'clock position. A positive value indicates a counter clockwise measurement, a negative value indicates a clockwise measurement.
V16	Angle of the last hole in a bolt circle. The value can be positive or negative as described for V15. When using scaling, V16 is the scaling factor for the X axis. V16 2.000 would scale all X axis dimensions two times the original size. V16 .5 would scale X axis dimensions one half size. When used in scaling, this value is always positive. This applies to V17 and V18 when scaling is used.
V17	Number of holes to be drilled in a bolt circle. This is not necessarily the total number of holes in a 360 degree pattern, it is the number of holes to be machined. This value is always a positive whole number. When using scaling, V17 is the scaling factor for the Y axis.
V18	Diameter of bolt circle, or hole mill cycle. This value is always positive. When using scaling V18 is the scaling factor for the Z axis.
V20	Feedrate for the Z axis in G80 series canned cycles. This value is always positive.
V21	Drilling start height forthe Z axis in G80 series canned cycles. This value is always absolute and can be positive or negative. For G83 or G87, this value must be .100 above the work surface. This way, the first peck will always be into the workpiece and all pecks will be equal.
V22	Dwell time when using G82 or G89. The tool will dwell for a programmed time when the tool reaches the programmed depth. V22 .5 will cause a .5 second dwell when G82 or G89 is used. This value is always positive.
V23	Maximum peck when using G83 or G87. The control will make all pecks equal but will not exceed the maximum peck or the Z axis depth. This value is a positive incremental distance. This must be less than the total Z depth.
V40	Starting height in the Z axis for pocket milling. This must be .100 above the work surface. This value is always absolute and can be positive or negative.
V41	Length of the pocket in the X axis. When the center of the tool is positioned over the lower left corner, V41 is programmed as the total length of the pocket. This value is a positive

incremental distance. When circular pocket milling is used, V41 is the center of the pocket in the X axis. This can be positive or negative and is always an absolute value.

V42 Width of the pocket in the Y axis. This is a positive incremental distance. When circular pocket milling is used, V42 is the center of the pocket in the Y axis. This can be positive or negative and is always an absolute value. When helical interpolation is used, V42 is the number of complete 360° revolutions required between the starting position and ending position. If the XY positions are not the same for the start and end of the helix, a fractional part of a revolution will be made to achieve the programmed motion.

V43 Depth of the pocket in the Z axis. This value is absolute and can be positive or negative.

V44 Pocket corner radius. This must be entered even if equal to the tool radius. This is always a positive value. If circular pocket milling is used, V44 is the diameter of the circle.

V45 Stepover in the X and Y axis. The step over in a rectangular pocket should not exceed 70% of the tool diameter. This value is an incremental distance and can be positive or negative. A positive value will produce a climb mill cut for roughing, a negative value will produce a conventional cut during roughing.

V46 Maximum depth of cut in Z axis. This value is a positive incremental distance.

V47 Stock left for a finish cut. Material will be left on the sides and bottom for a finish cut. This value is a positive incremental distance.

V48 Finish pass feed rate. This value must be entered even if equal to the roughing feedrate. This value must be positive.

V49 Tool diameter. This is the actual tool diameter used. Cutter diameter compensation must not be active during a pocket milling cycle. This is a positive value.

SECTION 12:
ADVANCED SOFTWARE SYSTEM-TWO

NOTE:

The following canned cycles are standard with all Crusader II controls purchased after June 1, 1985. The Advanced Software System-Two is available as an optional package to owners of Crusader controls purchased before the above date.

The Advanced Software System-Two package gives the customer canned cycles for several intricate milling operations, such as Elliptical Interpolation, Spiral Rotation and Mold Rotation to name a few. The Advanced Software System-Two package combines exceptional power and versatility with the industry's easiest-to-program CNC controls.

A. ELLIPTICAL INTERPOLATION (G05):

This canned cycle can be used to cut a full ellipse or any portion of an ellipse. This canned cycle has built-in cutter compensation. Before programming with this cycle, the cutter must be in the starting position (if cutter compensation is being used, then the cutter must be in the compensated position).

All dimensions given while programming this cycle must be the actual ellipse dimensions and not compensated.

> **NOTE:**
> When cutting an ellipse, the control calculates dimensions to the center of the cutter which is a true ellipse. The length and width will be produced as programmed, but arcs will be larger on external and smaller on internal arcs. This must be taken into account when cutting an ellipse. The reason is a line drawn parallel to an ellipse is not an ellipse.

The following variables must be programmed when using the Elliptical Interpolation (G05) cycle:

V01: Direction of cutter movement (Clockwise (CW) is a minus direction; Counterclockwise (CCW) is a plus direction) and Plane Selection (XY=1, XZ=2, and YZ=3). For example: CW, XY plane equals -1.

V02: Incremental distance from starting point on ellipse to center of X-axis.

V03: Incremental distance from starting point of ellipse to center on Y-axis.

V04: Incremental distance from start to finish on X-axis.

V05: Incremental distance from start to finish on Y-axis.

V06: Length X.

V07: Width Y.

V49: Cutter diameter. A plus diameter means the cutter will cut on the outside of programmed dimensions. A minus diameter will cause the cutter to cut inside programmed dimensions. Zero diameter will cause the center line of the cutter to cut along the programmed path.

G05: Activate.

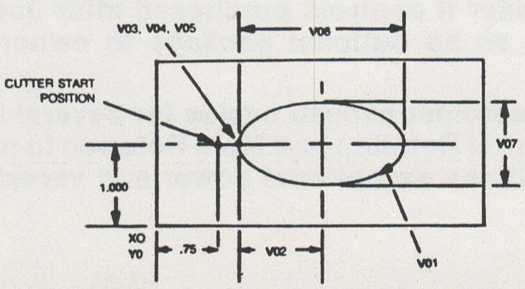


Figure 86 ELLIPTICAL MILLING CYCLE

ELLIPTICAL INTERPOLATION EXAMPLE PROGRAM:

1.	**X.75 Y1.**	Position cutter to starting point.
2.	**V01 -1.**	CW motion, X and Y plane.
3.	**V02 1.**	Start to center X.
4.	**V03 0**	Start to center Y.
5.	**V04 0**	Start to finish X.
6.	**V05 0**	Start to finish Y.
7.	**V06 2.**	Length X.
8.	**V07 1.**	Width Y.
9.	**V49 .5**	Cutter diameter.
10.	**G05**	

B. SPIRAL MILLING: PLANAR OR HELICAL (G06):

When activated by G06, this canned cycle can be used to mill either planar spirals (2 axis), or helical spirals (3 axis). The cutter must be at the actual Z depth and X, Y starting position before V01 is programmed.

The following variables must be used when programming with G06:

V01: Plane and direction of spiral (see explanation in section A).

V02: Incremental distance from start to X axis center.

V03: Incremental distance from start to Y axis center.

V04: Incremental distance from start to X axis finish point.

V05: Incremental distance from start to Y axis finish point.

V06: Incremental distance from start to Z axis finish point.

V07: Number of times the cutter crosses the line drawn through the center and start point (stopping on the line is not crossing it).

G06: Activate.

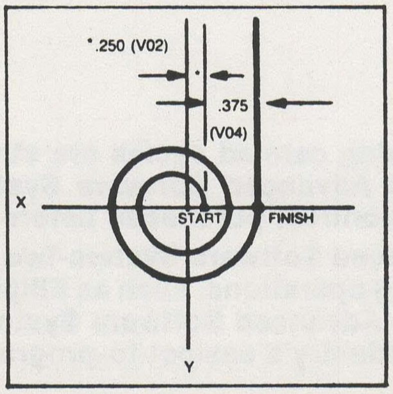


Figure 87 SPIRAL (PLANAR OR HELIX)

NOTE:
Cutter compensation cannot be used in this canned cycle.

SPIRAL MILLING EXAMPLE PROGRAM:

1.	**V01 +1**	X, Y CCW
2.	**V02 -.25**	Start to center X
3.	**V03 0**	Start to center Y
4.	**V04 .375**	Start to finish X
5.	**V05 0**	Start to finish Y
6.	**V06 -.060**	Start to finish Z
7.	**V07 1**	Number of complete turns.

C. CONICAL MILLING CYCLE (G48):

This cycle is used for cutting straight conical, concaved or convexed radial shapes. A ball end mill has to be used in order to produce a radius. All dimensions are programmed to the center of the cutter. The cutter must be in the X, Y and Z axis starting position before programming this cycle. The following variables must be programmed when using the G48 cycle:

V60: Angle at which the cone will be cut. The 3 o'clock position is Zero angle.
V62: Length of cylinder (center of cutter to center of cutter).
V63: Number of cycles. One cycle is equal to a cut in each direction across the cavity.
V64: Initial radius (radius to be cut minus the radius of the cutter).
V65: Final radius (radius to be cut minus the radius of the cutter).
V66: Concave: 0 Convex: 1
G48: Activate.

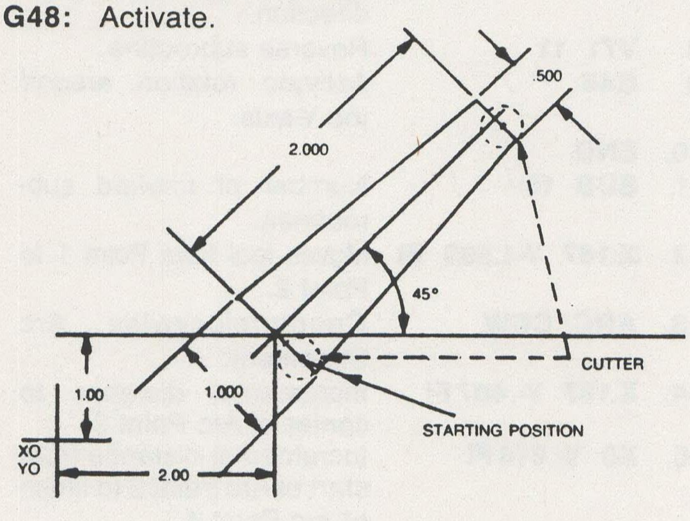


Figure 88 CONICAL MILLING CYCLE

CONICAL MILLING EXAMPLE PROGRAM:

The following program will cut the cavity shown in the figure below, using a 3/8″ ball end mill:

1.	**X2.221 Y.779**	Start position of cutter.
2.	**Z-.1875**	Z depth minus radius of cutter.
3.	**V60 45.**	Cone to be cut at 45 degrees CCW.
4.	**V62 2.000**	Length of cylinder.
5.	**V63 16**	Number of cycles. In this example, the cutter would take 32 passes across the cone or, 16 in each direction.
6.	**V64 .3125**	Initial radius. Radius to be produced is .5. Radius of cutter is .1875. Therefore, .5-.1875=.3125.
7.	**V65 .0625**	Final radius. Radius to be produced is .25. Radius of cutter is .1875. Therefore, .25-.1875=.0625.
8.	**V66 0**	Concaved cavity.
9.	**G48**	Activate.

NOTE:
In order to produce a half-circle, the Z depth must be set down into the surface by the distance of the radius of the cutter.

D. RADIAL CONICAL MILLING CYCLE (G49):

This cycle will cut a radial cone. The arc center around which the cone is to be cut must be programmed as X0 Y0. The cutter must be at the starting point of the cone. The Z axis starting depth must be the radius of the cutter and into the part in order to produce a full radius.

The following variables must be programmed when using the G49 cycle:

V62: Finish angle. The 3 o'clock position is equal to Zero degrees.
V63: Number of cycles. One cycle is 1 pass across in each direction.

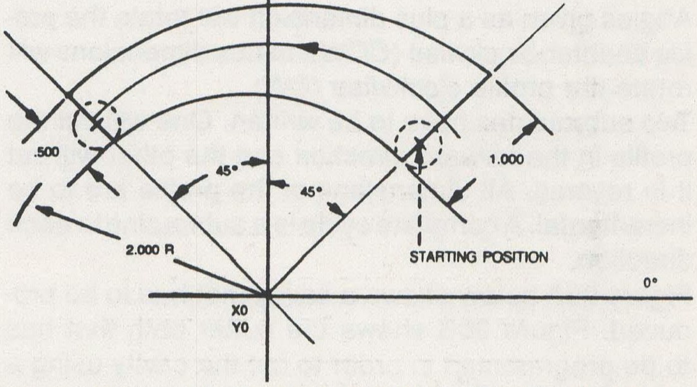


Figure 89 RADIAL CONICAL CAVITY MILLING

V64: Initial radius (starting radius minus cutter radius).
V65: Final radius (finish radius minus cutter radius).
V66: Concave: 0 Convex: 1
V79: Direction of arc to be cut (CW=-1.0 or CCW=1.0).
G49: Activate.

RADIAL CONICAL MILLING EXAMPLE PROGRAM:

1.	**X1.193 Y1.193**	Starting position of cutter.
2.	**Z-.1875**	Z depth to radius of cutter.
3.	**V62 135.**	Finish angle. Angle is 135 degrees CCW.
4.	**V63 16**	Number of cycles. This will actually take 32 passes, or 16 in each direction across the cavity.
5.	**V64 .3125**	Initial radius. Radius to be produced is .5. Cutter radius is .1875. Therefore, .5-.1875=.3125.
6.	**V65 .0625**	Final radius. Radius to be produced is .25. Cutter radius is .1875. Therefore, .25-.1875=.0625.
7.	**V66 0**	Concaved cavity.
8.	**V79 1.0**	Cavity is to be cut in a counterclockwise rotation.
9.	**G49**	Activate.

E. MOLD ROTATION CYCLES (G45, G46 AND G47):

These canned cycles allow the operator to cut a profile and then rotate it around an axis.

G45: Rotates around the X axis.
G46: Rotates around the Y axis.
G47: Rotates around the Z axis.

Center variables have to be programmed to make these cycles work. After the profile is programmed then this position is considered to be Zero degrees. Angles given as a plus dimension will rotate the profile counter-clockwise (CCW); minus dimensions will rotate the profile clockwise (CW).

Two subroutines have to be written. One will cut the profile in the forward direction and the other will cut it in reverse. All dimensions of the profile are to be incremental. A complete cycle is a subroutine in each direction.

Figure 90A below shows a cavity that has to be produced. Figure 90B shows the cutter path that has to be programmed in order to cut the cavity using a 3/8″ ball end mill.

The following variables must be defined when using this canned cycle:

V61: Starting angle.
V62: Finishing angle.
V63: Number of complete cycles. One cycle is equal to one pass in each direction.
V70: Forward subroutine (first pass of the cycle).
V71: Reverse subroutine (second pass of the cycle).
G46: Activate rotation around Y-axis.

NOTE:
Remember that V70 and V71 above will combine to make one complete cycle.

MOLD ROTATION EXAMPLE PROGRAM:

1.	**X0 Y0 RA**	Moves tool to starting position.
2.	**Z.2 RA**	Moves tool down to top of part.
3.	**Z0 FA**	Feed tool to starting depth.
4.	**V61 0**	Starting angle.
5.	**V62 -180**	Finish angle rotation is 180 degrees clockwise, therefore a minus symbol is used.
6.	**V63 9**	Number of complete cycles. The tool will take 18 passes, so the number of cycles indicated for V63 equals 9.
7.	**V70 10**	Forward subroutine. This is the subroutine in forward direction.
8.	**V71 11**	Reverse subroutine.
9.	**G46**	Activate rotation around the Y-axis.
10.	**END**	
11.	**SUB 10**	Number of forward subroutines.
12.	**X.167 Y-1.593 FI**	Moves tool from Point 1 to Point 2.
13.	**ARC CCW**	Counterclockwise Arc Statement.
14.	**X.167 Y-.407 FI**	Incremental distance to center of Arc Point 3.
15.	**X0 Y-.814 FI**	Incremental distance from start of Arc Point 2 to finish of Arc Point 4.
16.	**ARC CCW**	Arc End Statement.
17.	**X.167 Y-1.593**	Incremental distance from Point 4 to Point 5.

```
18. AUX 9030
19. X0 Y-4. Z0 FA        Reposition move.
20. END                  End of Subroutine.
21. SUB 11               Number of subroutines in
                         reverse direction.
22. X-.167 Y1.593 FI     Incremental move from
                         Point 5 to Point 4.
23. ARC CW
24. X.167 Y.407 FI       Incremental distance from
                         Point 4 to center of Arc
                         Point 3.
25. X0 Y.814 FI          Incremental distance from
                         Point 4 to Point 2. End of
                         Arc.
26. ARC CW               Arc End Statement.
27. X.167 Y1.593 FI      Incremental distance from
                         Point 2 to Point 1.
28. AUX 9030
29. X0 Y0 Z0 FA          Reposition move.
30. END                  End of subroutine.
```

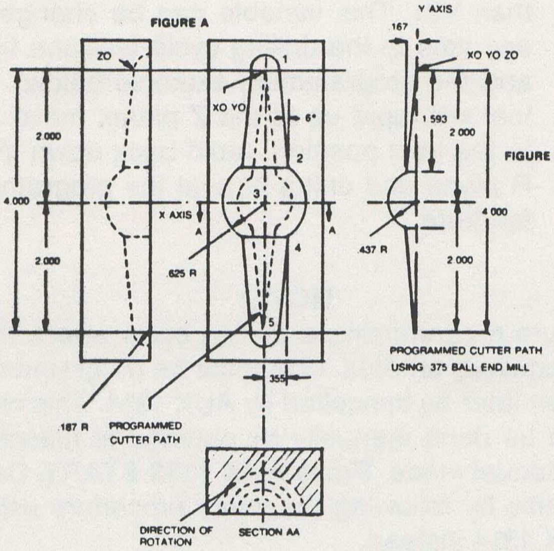


Figure 90 MOLD ROTATION

F. CANCELLING PLANE ROTATION (AUX 9030):

This AUX code (9030) is used to cancel AUX codes 9031, 9032 and 9033.

G. PLANE ROTATION (AUX 9031, 9032 AND 9033):

These AUX codes are used to rotate a profile or shape around an axis:

AUX 9031: Rotates around the X axis.
AUX 9032: Rotates around the Y axis.
AUX 9033: Rotates around the Z axis.

The point from which the profile or shape is rotating must be considered "0" during programming. The profile must be programmed incrementally. Also, please note that cutter compensation cannot be used with the Plane Rotation cycle.

The figure below shows a shape cut using AUX 9033 (Z plane rotation). Only the first 45-degree segment was actually programmed. See the programming example that follows.

PLANE ROTATION EXAMPLE PROGRAM (ONE):

```
1.  TOOL 1001
2.  Z (Offset) RA
3.  TOOL 0
4.  Z0 RA
5.  TOOL 1
6.  X0 Y1.875 RA     Move to start position.
7.  Z-.1 RA          Move tool down.
8.  FEED 10.
9.  Z-.1 FA
10. CALL 1           Call subroutine number one.
11. V60-45.          Angle profile to be rotated.
12. CALL 2           Call subroutine number two.
13. V60-90.          Angle of second rotation.
14. CALL 2
15. V60-135.         Angle of third rotation.
16. CALL 2
17. V60-180.         Angle of fourth rotation.
18. CALL 2
19. V60-225.         Angle of fifth rotation.
20. CALL 2
21. V60-270.         Angle of sixth rotation.
22. CALL 2
23. V60-315.         Angle of seventh rotation.
24. CALL 2
25. AUX 9030         Cancel rotation.
26. Z.1 FA           Retract cutter from part.
27. TOOL 0
28. Z0 RA
29. END
```

Subroutine One: Part Profile

```
30. SUB 1
31. ARC CW
32. X0 Y-.375 FI
33. X.3465 Y-.5185 FI
34. ARC CW
35. ARC CCW
36. X.1838 Y-.0762 FI
37. X.3677 Y-.1523 FI
38. ARC CCW
39. ARC CW
40. X.3465 Y-.1435 FI
41. X.6116 Y-.1216 FI
42. ARC CW
43. END
```

Subroutine Two: Plane Rotation

44. **SUB 2**
45. **AUX 9033** Activates Z-plane rotation.
46. **CALL 1** Calls Subroutine One.
47. **AUX 9030** Cancels plane rotation.
48. **END**

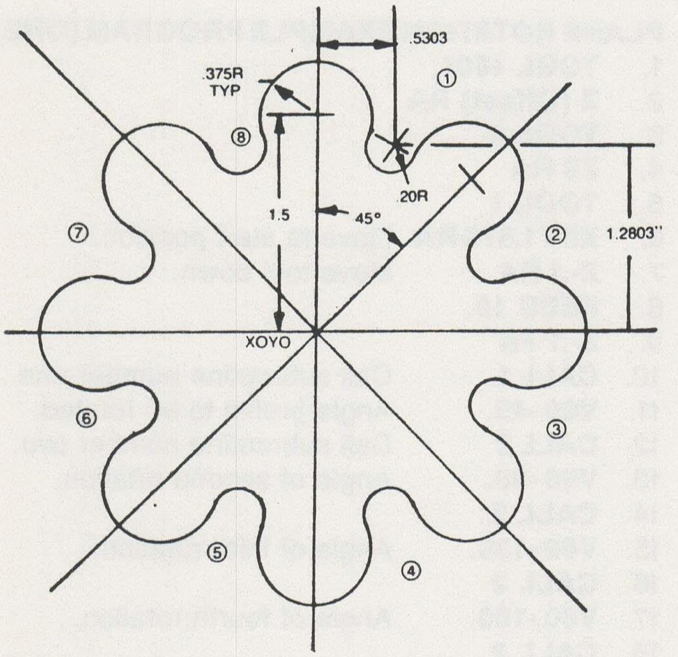


Figure 91 PLANE ROTATION

PLANE ROTATION EXAMPLE PROGRAM (TWO):

Another example of plane rotation would be to rotate a pocket milling cycle. To do this, Zero must be set at the bottom left corner of the pocket (if this point is not Part Zero, then a Zero Shift can be used). The following program will rotate a 2″ x 1″ pocket 45-degrees:

1. **TOOL 1001**
2. **Z (Offset) RA**
3. **TOOL 0**
4. **Z0 RA**
5. **X0 Y0 RA**
6. **TOOL 1**
7. **FEED 10.**
8. **V40 .1**
9. **V41 2.**
10. **V42 1.**
11. **V43 -.25**
12. **V44 .25**
13. **V45 .175**
14. **V46 .25**
15. **V47 .010**
16. **V48 20.**
17. **V49 .25**
18. **V50 0**
19. **V60 45.**
20. **AUX 9033**
21. **G78**
22. **AUX 9030**
23. **TOOL 0**
24. **Z0 RA**
25. **END**

H. DRILLING CANNED CYCLES (G80 through G89)

The following variables are used in describing tool movement for the Drilling Canned Cycles (G80 through G89):

V20: Specifies the feedrate for Z axis moves in drilling canned cycles.

V21: Specifies the height at which the tool will start feeding at the programmed feedrate. This must be absolute (R plane).

V24: Specifies the height the tool will retract to before the next move in X and Y. This dimension must be absolute and cannot be less than V21. This variable can be changed at any time in the drilling cycle (see the figure and the programming example below). The tool will rapid up to the Z plane, move over to the next position, rapid back down to the R plane and drill a hole at the programmed feedrate.

NOTE:

Before programming a drilling cycle where V24 is required, an AUX 1313 must be programmed. It can later be cancelled by AUX 1314. This may also be done manually by putting the machine in Manual mode. Press **AUX 1313 START.** Cancel this by following the same procedure using AUX 1314 instead.

DRILLING CANNED CYCLE EXAMPLE PROGRAM:

This example program uses G81 and changes V24.

1. **AUX 1313**
2. **V20 10.** Feedrate.
3. **V21 .1** R retract.
4. **V24 .5** Z retract.
5. **G81** Drilling cycle.
6. **Z-.5** Z drilling depth.
7. **X1.0 Y1.0**
8. **V24 .1** Z retract will change after next hole.
9. **X3.0**
10. **Y3.0**

11.	**V24 .5**	Z retract will change after next hole.
12.	**X1.0**	
13.	**G80**	Cancels drilling cycle.
14.	**AUX 1314**	Cancels AUX 1313.

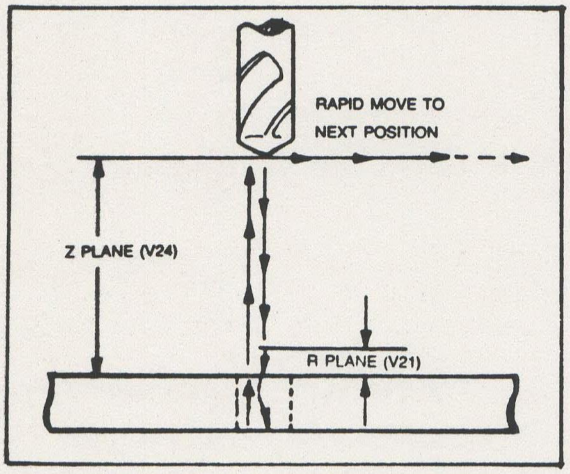


Figure 92 DRILLING CANNED CYCLES

SHEET ______ OF ________

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # ____________________ PROGRAMMER ____________________ DATE ______________

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1												
2												
3												
4												
5												
6												
7												
8												
9												
0												
1												
2												
3												
4												
5												
6												
7												
8												
9												
0												

SHEET ______ OF ________

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # ____________________ PROGRAMMER ____________________ DATE ______________

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1												
2												
3												
4												
5												
6												
7												
8												
9												
0												
1												
2												
3												
4												
5												
6												
7												
8												
9												
0												

Anilam Electronics Corporation 5625 N.W. 79th Avenue, Miami, Florida 33166 / (305) 592-2727 / Telex 26-4024

SHEET ______ OF ________

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # ____________________ PROGRAMMER ____________________ DATE ______________

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1												
2												
3												
4												
5												
6												
7												
8												
9												
0												
1												
2												
3												
4												
5												
6												
7												
8												
9												
0												

Anilam Electronics Corporation 5625 N.W. 79th Avenue, Miami, Florida 33166 / (305) 592-2727 / Telex 26-4024

SHEET ______ OF ________

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # ____________________ PROGRAMMER ____________________ DATE ______________

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1												
2												
3												
4												
5												
6												
7												
8												
9												
0												
1												
2												
3												
4												
5												
6												
7												
8												
9												
0												

Anilam Electronics Corporation 5625 N.W. 79th Avenue, Miami, Florida 33166 / (305) 592-2727 / Telex 26-4024

SHEET ______ OF ________

ANILAM CRUSADER PROGRAM SHEET

PART OR DRWG. # ____________________ PROGRAMMER ____________________ DATE ______________

TOOL LIST	OFFSETS	SETUP INSTRUCTIONS

EVENT NO.	X	Y	Z	RAPID /FEED	INCR /ABS	ARC CW CCW	G	V	TOOL	DWELL /AUX	DO/ CALL	SUB/ END
1												
2												
3												
4												
5												
6												
7												
8												
9												
0												
1												
2												
3												
4												
5												
6												
7												
8												
9												
0												

Missing

(2) Cassettes

(1) Cassette player cable

~~Analam 1/800/327-6340~~

5/19/93 Jerry Bouvier X 304 (305) 592-2727

1. Drop out may be caused by vibration of Z axis switches. Replaced 3 amp switches w/ cherry 5 amp switches

Bill Grunwald
X 303

5/20/93